广州数控设备有限公司
GSK CNC EQUIPMENT CO., LTD.

Add: No.7, 1st. Street, Luochong North Road, Luochongwei, Guangzhou, 510165, China

Http://www.gsk.com.cn E-mail:sale1@gsk.com.cn

Tel: 86-20-81796410/81797922 Fax: 86-20-81993683

All specifications and designs are subject to change without notice.

Temporary Edition 10th Jun. 2010

HTTP://WWW.GSK.COM.CN E-MAIL:SALE1@GSK.COM.CN

GSK 980MDa Milling CNC System User Manual

GUANGZHOU
CHINA

USER MANUAL

GSK 980MDa Milling CNC System

广州数控设备有限公司
GSK CNC EQUIPMENT CO., LTD.

This user manual describes all items concerning the operation of this CNC system in detail. However, it is impossible to give particular descriptions for all unnecessary or unallowable operations due to length limitation and products application conditions；Therefore, the items not presented herein should be considered impractical or unallowable.

Copyright is reserved to GSK CNC Equipment Co., Ltd. It is illegal for any organization or individual to publish or reprint this manual. GSK CNC Equipment Co., Ltd. reserves the right to ascertain their legal liability.

Preface

Your Excellency,

We are honored by your purchase of products from GSK CNC Equipment Co., Ltd.

This manual introduces programming, operation and connection of GSK980MDa CNC Milling Machine in detail. To ensure safe and efficient work, please read this manual carefully before installation and operation.

Warning and Precaution

Accident may occur by improper connection and operation！This system can only be operated by authorized and qualified personnel.

Please read this manual carefully before operation！

Special caution:

The power supply fixed on/in the cabinet is exclusively used for the CNC system made by GSK.

It can't be applied to other purposes, or else it may cause serious danger.

This manual is reserved by end user.

Cautions

■ Transportation and Storage

- Packing box over 6 layers in pile is not allowed.
- Never climb the packing box, neither stand on it, nor place heavy objects on it.
- Do not move or drag the products by the cables connected to it.
- Forbid collision or scratch to the panel and display screen.
- Avoid dampness, insolation and drenching.

■ Open-package Inspection

- Confirm that the products are the required ones.
- Check that the products are not damaged in delivery.
- Confirm that the parts in packing box are in accordance with the packing list.
- Contact us in time if any inconsistence, shortage or damage is found.

■ Wiring

- Only qualified personnel can connect the system or check the connection.
- The system must be earthed, and the earth resistance must be less than 0.1Ω. The earth wire cannot be replaced by a neutral wire (zero wire).
- The connection must be correct and firm to avoid any fault or unexpected consequence.
- Connect with surge diode in the specified direction to avoid damage to the system.
- Switch off power supply before plugging out or opening electric cabinet.

■ Troubleshooting

- Cut off the power supply before troubleshooting or component replacement.
- Check for fault when short circuit or overload occurs. Restart can only be done after troubleshooting.
- Frequent switching on/off of the power is forbidden, and the interval time should be at least 1 min.

ANNOUNCEMENT!

- **This manual describes various possibilities as much as possible. However, operations allowable or unallowable cannot be explained one by one due to so many possibilities that may involve with, so the contents that are not specially stated in this manual shall be considered as unallowable.**

WARNING！

- **Please read this manual and a manual from machine tool builder carefully before installation, programming and operation, and strictly observe the requirements. Otherwise, products and machine may be damaged, workpiece be scrapped or the user be injured.**

NOTE！

- **Functions, technical indexes (such as precision and speed) described in this user manual are only for this system. Actual function deployment and technical performance of a machine tool with this CNC system are determined by machine tool builder's design, so functions and technical indexes are subject to the user manual from machine tool builder.**
- **Though this system is employed with integrated operator panel, the functions of the keys on the panel are defined by PLC program (ladder diagram). It should be noted that the keys functions described herein are for the standard PLC program (ladder diagram).**
- **Refer to the user manual from machine tool builder for function and meaning of keys on control panel.**

This manual is subject to change without further notice.

Volume Ⅰ Programming

Introduces product specification, types, command codes and format of programs.

Volume Ⅱ Operation

Describes the operation methods of GSK980MDa CNC Milling Machine.

Volume Ⅲ Installation

Describes the methods for installation, connection and setting of GSK980MDa CNC Milling Machine.

Appendix

Describes standard ladder diagram functions and lists the alarm codes.

Safety Responsibility

Manufacturer Responsibility

——Be responsible for the danger which should be eliminated and/or controlled on design and configuration of the provided CNC systems and accessories.

——Be responsible for the safety of the provided CNC systems and accessories.

——Be responsible for the provided information and advice for the users.

User Responsibility

——Be trained with the safety operation of CNC system and familiar with the safety operation procedures.

——Be responsible for the dangers caused by adding, changing or altering to the original CNC systems and the accessories.

——Be responsible for the failure to observe the provisions for operation, adjustment, maintenance, installation and storage in the manual.

Contents

Volume Ⅱ OPERATION

VOLUME Ⅲ INSTALLATION

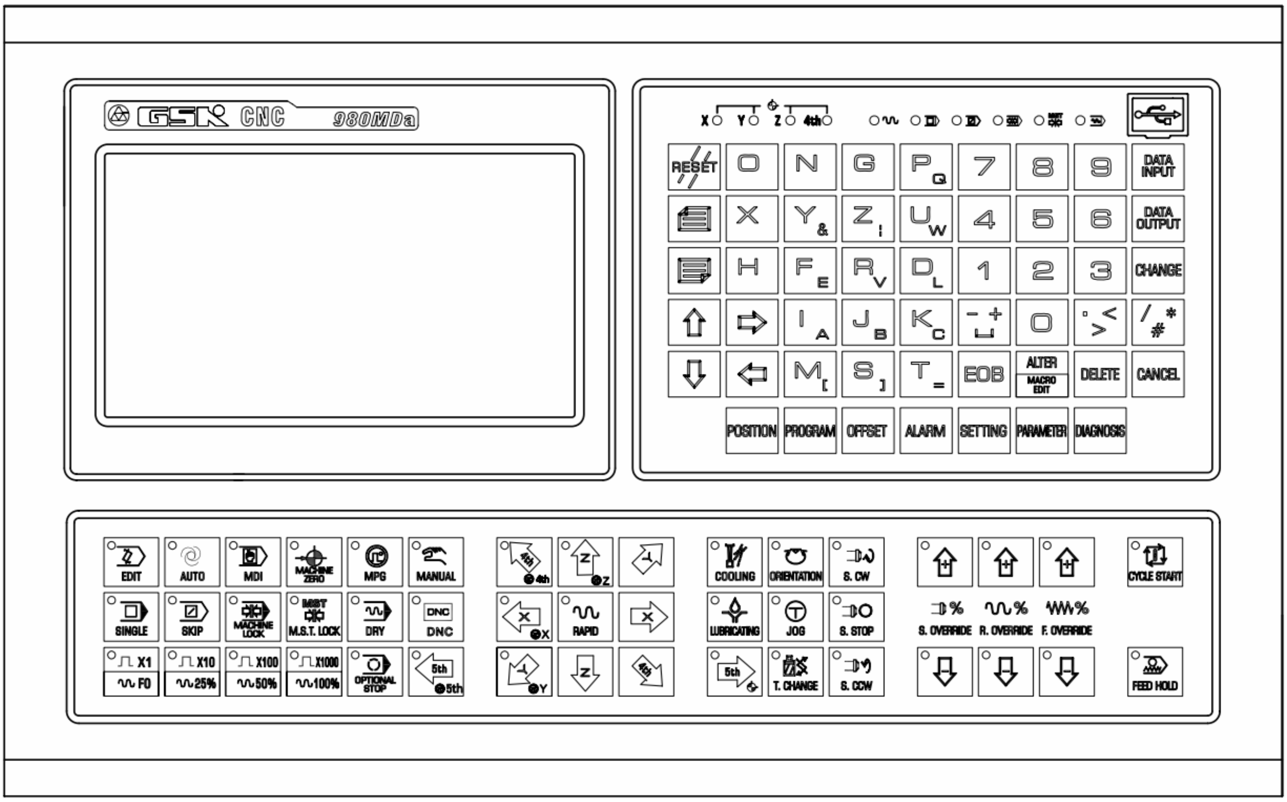
APPENDIX

VOLUME I PROGRAMMING

CHAPTER 1 PROGRAMMING FUNDMENTALS

1.1 Introduction

GSK980MDa Milling Machine is a new generation of CNC system developed by GSK Company. As the upgraded version of GSK980MD, it supports milling, boring and drilling cycle. It employs 32 bits high-capability CPU and very large scale programmable device FPGA, applies real-time multi-task control technology and hardware interpolation technology, and is able to perform μm level precision motion control and PLC logic control. GSK980MDa is the optimum choice for upgrading CNC milling machine.



Characteristics：

- ✓ Five axes control (X, Y, Z ,4th and 5th); 3 axes linkage; optional interpolation precision (1μm/0.1μm); maximum speed 60m/min; optional axis types (linear axis or revolving axis) for the 4th and 5th axes; CS axis control available for the 4th and 5th axes.
- ✓ Electronic gear ratio: (1～32767):(1～32767)
- ✓ Screw-pitch error compensation, backlash compensation, tool length compensation, tool abrasion compensation and tool nose radius compensation.
- ✓ Embedded with PLC can be downloaded to CNC from PC.
- ✓ DNC function supports for real-time program transmission for machining.
- ✓ Compatible with G commands in GSK980MC, GSK928MA and GSK980MD. 26 kinds of canned cycles, such as drilling/boring, circular/rectangular groove rough-milling, full circle/rectangular finish-milling, linear/rectangular/arc continuous drilling.
- ✓ Spindle encoder tapping and rigid tapping can be detected during tapping cycle, so that high precision machining can be performed.

- ✓ Metric/inch programming; automatic chamfering function and tool life management function.
- ✓ Chinese, English, Russian and Spanish display selected by the parameters.
- ✓ Full screen program editing; 40MB program capacity for storing up to 40000 of part programs.
- ✓ USB data communication; CNC system upgrading, machining programs reading through U disk and bidirectional transfer between CNC and U disk.
- ✓ Alarm log; multi-level passwords for equipment maintenance and management.
- ✓ Bidirectional transfer between CNC and CNC, CNC and PC; upgrade of CNC software and PLC programs;
- ✓ The installation dimensions and the electric ports are compatible with GSK980MD, GSK980MC.

Specifications

Motion control	Controlled axes: five axes (X,Y,Z,4th and 5th); (for the 4th and 5th axes) optional axis types (linear axis or revolving axis) and CS contouring control available;
	Interpolation functions: linear interpolation (for X, Y, Z, 4th and 5th axes); helical interpolation (for X, Y and Z axes); circular interpolation (for arbitrary 2 axes).
	Position command range: -99999999～99999999; least command increment: 1μm/0.1μm; (selected via parameters)
	Electronic gear ratio: command multiplier 1～32767, command frequency divisor 1～32767
	Rapid traverse speed: maximum 60000mm/min Rapid traverse override: F0, 25%, 50%, 100% four levels real-time tuning
	Cutting feedrate: maximum 15000mm/min (feed per min.) or 500mm/r. (feed per rotation) Feedrate override: 0～150% sixteen-level real-time tuning
	Manual feedrate: 0～1260mm/min sixteen-level real-time tuning
	MPG feed: 0.001, 0.010, 0.100,1.000mm four gears.
	Acceleration/deceleration type: S-type for rapid traverse; exponential-type for cutting feed.
	Automatic chamfering
G Code	65 kinds of G codes：G00, G01, G02, G03, G04, G10, G11, G17, G18, G19, G20, G21, G28, G29, G30, G31, G40, G41, G42, G43, G44, G49, G54, G55, G56, G57, G58, G59, G65, G66, G67, G73, G74, G80, G81, G82, G83, G84, G85, G86, G88, G89, G90, G91, G92, G94, G95, G98, G99, G110, G111, G112, G113, G114, G115, G134, G135, G136, G137, G138, G139, G140, G141, G142, G143
Macro command	31 kinds of arithmetic, logical operations and skip can be achieved by macro command G65
	Macro statement command. eg:IF,WHILE,GOTO
Operation mode	Seven operation modes: EDIT, AUTO, MDI, DNC, MACHINE ZERO, MPG/STEP and MANUAL.
Tapping	Tapping function: lead 0.001～500mm or 0.06～25400 pitch/inch

	Encoder tapping: settable line number of encoder（0 or100p/r～5000p/r）; no detect for spindle encoder (when the line number is set to 0)
	Rigid tapping: by rotary axis
	Drive ratio between encoder and spindle:（1～255）:（1～255）
Precision compensation	Backlash compensation: 0～2.000mm
	Pitch error compensation: 255 compensation points per axis; compensation amount of each point: ±0.255mm.
	Tool compensation: 32 groups tool length compensation, tool wear compensation, cutter compensation C
M command	Special M commands (redefinition unallowed): M02,M29, M30, M98, M99,M9000～M9999. Other M □□ commands are defined or disposed by PLC program.
	M commands defined by standard PLC program: M00, M03, M04, M05 M08, M09, M10, M11, M32, M33
T command	tool number T01～T32 (32 numbers at most); manual tool change or auto-tool change selected by the parameters; auto tool change sequence set by PLC program.
	Tool life management; 32 groups, 8 kinds/groups of tool life management data
Spindle speed control	Speed switching value control: S □□ command is defined or disposed by PLC program; the standard PLC programs S1, S2, S3 and S4 directly output; The output of S1,S2, S3, and S4 are closed by S0.
	Speed analog voltage control: the spindle speed per minute commanded by S codes; output 0～10V voltage to spindle converter; spindle stepless speed changing supports 4 spindle mechanical gears
PLC function	9 kinds of basic commands; 23 kinds of function commands; 2-level PLC program involving up to 5000 steps (2μs processing time for each step). 8ms refresh cycle for the first level program; Ladder diagram edit software and communication software downloadable
	Integrated machine panel: 44 points input (key), 44 points output (LED) Basic I/O: 41 points input/ 36 points output
Display interface	Displayer: 480×234 lattice, 7'' wide-screen multi-color LCD,
	Display modes: Chinese, English, Russian, Spanish display selected by parameters; machining path displayable
Program edit	Capacity: 40MB for up to 40000 part programs; custom macro program call; 4 nesting-levels of subprogram
	Edit modes: full-screen editing; absolute/incremental programming
USB	CNC system upgrade
	Part programs reading in USB
	Bidirectional files transfer between CNC and USB (including programs, parameters, PLC backup and recovery)
Clock display	Clock, date and week display.
Serial Communication	bidirectional transfer between CNC and PC, CNC and CNC (involving programs, parameters, tool compensation data); download and upgrade of system software and PLC program serial ports

Matching drive unit	AC servo or step drive device by using the pulse+direction signal input. (DA98 or DY3 series)

G Code Table

Code	Function	Code	Function	Code	Function
G00	Positioning (rapid traverse)	*G54	Workpiece coordinate system 1	G92	Coordinate system setting
*G01	Linear interpolation	G55	Workpiece coordinate system 2	*G94	Feed per min.
G02	Circular/helical interpolation (CW)	G56	Workpiece coordinate system 3	G95	Feed per rotation
G03	Circular/helical interpolation (CCW)	G57	Workpiece coordinate system 4	*G98	Return to initial plane in canned cycle
G04	Dwell, exact stop	G58	Workpiece coordinate system 5	G99	Return to R point in canned cycle
G10	Tool life management	G59	Workpiece coordinate system 6	G110	Inner circle groove roughing (CCW)
G11	Tool life management end	G65	Macro program/ macro code	G111	Inner circle groove roughing (CW)
*G17	XY plane selection	G66	Macro program modal call	G112	Inner circle finishing (CCW)
G18	ZX plane selection	*G67	Macro program modal call cancel	G113	Inner circle finishing (CW)
G19	YZ plane selection	G73	High-speed peck drilling	G114	Circular outer finish milling (CW)
G20	Inch input	G74	Counter tapping cycle	G115	Outer circle finishing (CCW)
G21	Metric input	*G80	Canned cycle cancel	G134	Rectangular groove roughing (CCW)
G28	Reference position return	G81	Drilling cycle (spot drilling cycle)	G135	Rectangular groove roughing (CW)
G29	Return from reference position	G82	Drilling cycle (stepped hole boring cycle)	G136	Rectangular groove inner finishing (CCW)
G30	2nd, 3rd, 4th, reference position return	G83	Peck drilling cycle	G137	Rectangular groove inner finishing (CW)
G31	Skip function	G84	Tapping cycle	G138	Rectangular outer finishing (CCW)
*G40	Cutter compensation cancel	G85	Boring cycle	G139	Rectangular outer finishing (CW)
G41	Cutter compensation left	G86	Drilling cycle	G140	Rectangular continuous drilling (CW)
G42	Cutter compensation right	G88	Boring cycle	G141	Rectangular continuous drilling (CCW)

G43	Tool length compensation + direction	G89	Boring cycle	G142	Arc continuous drilling (CW)
G44	Tool length compensation – direction	*G90	Absolute programming	G143	Arc continuous drilling (CCW)
*G49	Tool length compensation cancel	G91	Incremental programming		

Note: mark " * " means initial state.

PLC Codes List

Code	Function	Code	Function	Code	Function
LD	Normal open contact read	SET	Setting	SPE	Subprogram end
LDI	Normal closed contact read	RST	Resetting	ADDB	Binary addition
OUT	Output coil	CMP	Comparison setting	SUBB	Binary subtraction
AND	Normal open contact in series	CTRC	Counter	ALT	Alternative output
ANI	Normal closed contact in series	TMRB	Timer	DIFU	Differential up
OR	Normal open contact in parallel	CODB	Binary code transformation	DIFD	Differential down
ORI	Normal closed contact in parallel	ROTB	Binary rotational control	MOVE	Logical AND
ORB	Serial block in parallel	MOVN	Data copy	PARI	Parity check
ANB	Parallel block in series	DECB	Binary decode	LBL	Program skip numbering
END1	first level program end	JMPB	Jump	CALL	Subprogram call
END2	Second level program end	SP	Subprogram numbering		

1.2 Program Execution

1.2.1 Program Execution Sequence

The current program can only be run in automatic mode. GSK980MDa cannot run more than 1 program at the same time, so only one program can be performed at a time. The cursor is ahead of the first block when a program is opened, and can be moved in EDIT mode. In automatic mode, when the machine is in stop state, the cycle start signal ( key on the panel or external cycle start signal) enables the program to be run from the block where the cursor is located. Usually, blocks are executed in sequence programmed in advanced. Program stops running till M02 or M30 is executed. The cursor

moves along with program execution. The program execution sequence or state will be changed in following conditions:

- Program running stops when  key or the Emergency Stop button is pressed;

  Program running stops when the CNC alarm or PLC alarm occurs;

- When the system is switched in EDIT or MDI mode, program stops running after the current block is executed. After switching to automatic mode again, when  key on the panel is pressed or external cycle start signal is ON, the program runs from the block where the cursor is located.

- If the operation mode is switched to MANUAL/MPG/STEP/MACHINE ZERO RETURN mode when the program is running, the execution dwells; after switching to automatic mode again, when  key on the panel is pressed or external cycle start signal is ON, the program runs from where it stops.
- The execution dwells when  key is pressed or external pause signal is cut off; program starts running from where it stops when  key on the panel is pressed or external cycle start signal is ON;
- The program dwells at the end of each block when the single block switch is on; after pressing key or switching on external cycle signal, program continuously runs from the next block;
- Blocks with mark “/” is skipped when the skip switch is ON.
- The object block is executed when command G65 or macro program skip (GOTO) is specified.
- When M98 or M9000~M9999 command is performed, the corresponding subprogram or macro program is called; M99 is executed at the end of the subprogram or macro program, after returning to the main program, the subsequent block (the one after the block in which the subprogram is called) is executed. (return to a specified block, if it is commanded by M99);
- When M99 command is specified in the middle of a main program which is not called by other programs, the current program is repeatly executed after returning to the head of the program.

1.2.2 Word Execution Sequence within Block

When multiple words (such as G, X, Y, Z, F, R, M, S, T,) are in one block, most of M, S, and T words are interpreted by NC and sent to PLC for processing. Other words are processed by NC directly. M98, M99, M9000~M9999 and S word (which specify the spindle speed in r/min, m/min) are directly processed by NC as well.

When G words share the same block with M00, M01, M02 and M30, M words are executed after G words, and NC sends corresponding signals to PLC for processing.

When the G words share the same block with the M98, M99, M9000~M9999, these M words are performed by NC after G words (the M signal not sent to PLC).

When G words and M, S, T words share the same block, PLC program (ladder diagram) determines the execution consequence (executed at the same time or G words before M, S, T words). Refer to the manual from tool builder for relevant words execution sequence.

1.3 Basic Axes Increment System

The increment system consists of the least input increment (for input) and least command increment (for output). The least input increment is the minimum unit for programming moving distance. The least command increment is the minimum unit for moving the tool on the machine. Both increments are represented in mm,inches.or deg.

The basic axes herein means X, Y, Z axes. The basic increment system includes IS-B and IS-C types which can be selected by bit ISC of parameter NO.038.

038	ISC							

ISC =1：The increment system is IS-C(0.1U)；

=0：The increment system is IS-B(1U)

In different increment system, different pulse output type enables different output speed. (Selected by bit ABPx of parameter NO.039)

039				ABP5	ABP4	ABPZ	ABPY	ABPX

ABPx =1：The impulse mode of axis is AB phases;

=0：The impulse mode of axis is impulse and direction.

1.3.1 Speed of Increment Systems

Output mode	**Speed**			
	1 u（IS-B）		**0.1u（IS-C）**	
	Metric machine system (mm/min)	Inch machine system (inch/min)	Metric machine system (mm/min)	Inch machine system (inch/min)
Pulse + direction	60,000	6,000	6,000	600
AB quadrature phase	240,000	24,000	24,000	2,400

1.3.2 Unit of Increment Systems

In different increment system, the least input/output increment varies with metric/inch system. The specific data is shown as follows:

1 u（IS-B）		**Least input increment (for input)**	**Least command increment (for output)**
Metric machine system	Metric input (G21)	0.001 (mm)	0.001 (mm)
		0.001 (deg)	0.001 (deg)
	Inch input (G20)	0.0001 (inch)	0.001 (mm)
		0.001 (deg)	0.001 (deg)
Inch machine	Metric input (G21)	0.001 (mm)	0.0001 (inch)
		0.001 (deg)	0.001 (deg)

system	Inch input (G20)	0.0001 (inch)	0.0001 (inch)
		0.001 (deg)	0.001 (deg)

0.1u（IS-C）		**Least input increment (for input)**	**Least command increment (for output)**
Metric machine system	Metric input (G21)	0.0001 (mm)	Metric machine system
		0.0001 (deg)	
	Inch input (G20)	0.00001 (inch)	
		0.0001 (deg)	
Inch machine system	Metric input (G21)	0.0001 (mm)	Inch machine system
		0.0001 (deg)	
	Inch input (G20)	0.00001 (inch)	
		0.0001 (deg)	

Least input increment (for input) is metric or inch can be set by G20 or G21.

Least command increment (for output) is metric or inch is determined by machine tool and set by bit SCW of parameter NO.004.

1.3.3 Data Ranges of Increment System

Limited by pulse output frequency, the data ranges may vary due to different increment system.

Increment system		**Command data input ranges**	**Data format**
1 u（IS-B）	Metric input (G21)	-99999.999 ~ 99999.999 (mm) -99999.999 ~ 99999.999 (deg)	5.3 5.3
	Inch input (G20)	-9999.9999 ~ 9999.9999 (inch) -9999.999 ~ 9999.999 (deg)	4.4 4.3
0.1u（IS-C）	Metric input (G21)	-9999.9999 ~ 9999.9999 (mm) -9999.9999 ~ 9999.9999 (deg)	4.4 4.4
	Inch input (G20)	-999.99999 ~ 999.99999 (inch) -999.9999 ~ 999.9999 (deg)	3.5 3.4

Note：5.3 in the table above indicates 5 integers and 3 decimals. Other data are alike.

1.3.4 Data Ranges and Unit of Increment System

- **Speed parameter**

Machine tool types decide the units of linear axes speed, i.e. mm/min for metric machine system is; 0.1inch/min for inch machine system.

The range of linear axis speed parameter is codetermined by machine tool type and increment system.

For example: data parameter NO.070: upper limit of cutting feedrate.

Machine tool type	Increment system	Linear axis speed unit	Parameter range	Rotary axis speed unit
Metric machine system	1 u（IS-B）	mm/min	10~ 60000	deg/min
	0.1u （IS-C）		10~ 6000	
Inch machine system	1 u（IS-B）	0.1inch/min	5~60000	
	0.1u（IS-C）		5~6000	

As rotary axes are not involved in metric-inch interconversion, the rotation speed unit is always deg/min.

The switch between different increment systems may cause the excess of permitted running speed set by data parameter. Therefore, at the first power-on after switching, the system automatically modifies relevant speed parameters and gives an alarm.

- **Increment parameter**

The unit and range of linear axis speed parameter are codetermined by machine tool type and increment system.

For example: parameter NO135: X axis software limit.

Machine tool type	Increment system	Linear axis increment unit	Linear axis parameter range
Metric machine system	1 u（IS-B）	0.001mm	-99,999.999~ 99,999.999
	0.1u（IS-C）	0.0001 mm	-9,999.9999~ 9,999.9999
Inch machine system	1 u（IS-B）	0.0001inch	-9,999.9999~ 9,999.9999
	0.1u（IS-C）	0.00001 inch	-999.99999~ 999.99999

As rotary axes are not involved in metric-inch interconversion, the rotary axis increment parameter unit is determined by increment system types. The ranges of rotary axis increment parameters are the same as that of metric machine tool.

Machine tool type	Increment system	Rotation axis speed unit	Rotation axis parameter range
Metric, inch machine tool system	1 u（IS-B）	0.001deg	0~ 99999.999
	0.1u（IS-C）	0.0001 deg	0~ 9999.9999

- **Coordinate data（G54～G59）**

The unit of linear axis coordinate data is determined by metric/inch input system, namely, mm for metric system, inch for inch system.

The ranges of linear axis coordinate data are codetermined by metric/inch input system and increment system. It is the same as command data input ranges. Shown as follows:

Increment system		**Linear axis coordinate data range**
1 u（IS-B）	Metric input (G21)	-99999.999 ~ 99999.999(mm)
	Inch input (G20)	-9999.9999 ~ 9999.9999(inch)
0.1u（IS-C）	Metric input (G21)	-9999.9999 ~ 9999.9999(mm)
	Inch input (G20)	-999.99999 ~ 999.99999(inch)

As rotary axis is not involve in metric-inch interconversion, the unit of rotary axis coordinate data is deg. The ranges of rotary axis coordinate data is the same as linear axis coordinate data ranges in metric system.

Input type	**Increment system**	**Rotary axis coordinate data range**
Metric, inch input	1 u（IS-B）	-99999.999 ~ 99999.999（deg）
	0.1u（IS-C）	-9999.9999 ~ 9999.9999(deg)

- **Tool compensation data**

The unit of tool compensation data is determined by metric/inch input system, namely, mm for metric input, inch for inch input.

The range of tool compensation data is limited as 9999999, determined by inch input system and increment system. It is smaller than command data. Shown as follows:

Input type	**Increment system**	**Tool compensation data unit**	**Tool compensation data range**
Metric input (G21)	1 u（IS-B）	mm	±9999.999
	0.1u（IS-C）		±999.9999
Metric input (G21)	1 u（IS-B）	inch	±999.9999
	0.1u（IS-C）		±99.99999

- **Screw-pitch error compensation data**

The unit and range of linear axis screw-pitch error compensation data is codetermined by machine tool type and increment system.

Shown as following table:

Machine tool type	Increment system	Linear axis screw-pitch error compensation data unit	Linear axis screw-pitch error compensation data range
Metric tool machine system	1 u（IS-B）	0.001mm	-255～255
	0.1u（IS-C）	0.0001mm	-2550～2550
Inch tool machine system	1 u（IS-B）	0.0001inch	-255～255
	0.1u（IS-C）	0.00001inch	-2550～2550

Rotary axes are not involved in metric-inch conversion. The unit of rotary axes screw-pitch error compensation is determined by increment system. The range is the same as that of the metric machine tool.

Machine tool system	Increment system	Rotary axis screw-pitch error compensation unit	Rotary axis screw-pitch error compensation range
Metric, inch machine system	1 u（IS-B）	0.001deg	0～255
	0.1u（IS-C）	0.0001 deg	0～2550

- **Graphic setting data**

The maximum and minimum data ranges of X, Y, Z set by graph is in accordance with the command data ranges.

Increment system		Graphic setting X,Y,Z ranges
1 u（IS-B）	Metric input (G21)	-99999.999 ~ 99999.999 (mm)
	Inch input (G20)	-9999.9999 ~ 9999.9999 (inch)
0.1u（IS-C）	Metric input (G21)	-9999.9999 ~ 9999.9999 (mm)
	Inch input (G20)	-999.99999 ~ 999.99999 (inch)

1.3.5 The Units and Ranges of Program Address Values

- **Definition and ranges of the pitch ：**

	Code	1 μ（IS-B）	0.1μ（IS-C）	Unit
Input in metric（G21）	F	0.001~500.000	0.0001~500.00	mm/pitch [lead]
	I	0.06~25400	0.06~2540	Pitch[lead]/inch
Inch input（G20）	F	0.0001~50.00	0.00001~50.0	inch//pitch [lead]
	I	0.06~2540	0.06~254	Pitch[lead]/inch

- **Speed F definition**

G94: feed per minute, F unit: mm/min

G95: feed per rotation, F definition and ranges are as follows:

	1 μ（IS-B）	**0.1μ（IS-C）**	**Unit**
Metric input（G21）	0.001~500.000	0.0001~500.0000	mm/revolution
Inch input（G20）	0.0001~50.0	0.00001~50.0	inch/revolution

1.4 Additional Axes Increment System

In the least increment system (IS-B or IS-C), under the condition that the additional axes are not involved in simultaneous control and just used for separate motion (such as feeding), and the requirement for precision is not high, when the least increment is 0.01, the feedrate will be much faster, greatly increasing the efficiency. Therefore, the additional axes least increment system is not necessary to be in accordance with the current least increment system. To meet various requirements of users, the system adds optional function to least increment system.

Additional axes increment system is set by state parameter No.026, No.028. Shown as follows:

026	**A4IS1**	**A4IS0**		RCS4			ROS4	ROT4

A4IS1, A4IS0：Select increment system of 4th.

A4IS1	**A4IS0**	**Increment System of 4TH**	**Least input/output**
0	0	Same to the X, Y, Z	
0	1	IS-A	0.01
1	0	IS-B	0.001
1	1	IS-C	0.0001

028	**A5IS1**	**A5IS0**		RCS5			ROS5	ROT5

A5IS1, A5IS0：Selecte increment system of 5th.

A5IS1	**A5IS0**	**Increment System of 5TH**	**Least input/output**
0	0	Same to the X, Y, Z	
0	1	IS-A	0.01
1	0	IS-B	0.001
1	1	IS-C	0.0001

Note: the least input/output in the table above are described without considering the metric/inch system and rotation axes.

1.4.1 Additional Axes in Current Increment System

When IS-B or IS-C is selected, the speed and range of additional axes are the same as described in 1.3.

1.4.2 Additonal Axes in IS-A Increment System

When IS-A is selected, the maximum speed of additional axes can reach 100 times of that of IS-B and IS-C. The relevant data and parameters ranges are the same as that of the current basic axes increment system. (Refer to section 1.3)

CHAPTER 2 MSTF CODES

2.1 M Codes (Miscellaneous Function)

The M codes are composed by code address M and 1~2 or 4 digits after the codes M is used for controlling the program execution or outputting M code to PLC.

M □□□□

→ Codes value (00~99, 9000~9999，leading zero can be omitted)

→ Address

M98, M99 and M9000~M9999 are independently processed by CNC, and the M codes are not output to PLC.

The function of M29 is fixed, namely, to output M codes to PLC.

The M02 and M03 are defined as program END codes by NC, meanwhile it also outputs M codes to PLC for the I/O control (spindle OFF, cooling OFF control etc.).

The PLC program can not change the meaning of the above-mentioned codes when the M98, M99 and M9000~M9999 are regarded as program CALL codes and the M02 and M30 are regarded as program END codes. The codes of other M codes are all output to PLC program for specifying the code function; please refer to the manual issued by machine tool manufacturer.

One block only has one M code. The CNC alarm occurs when two or more M codes are existed in one block.

Table 2-1 M code table for program execution

Codes	Functions
M02	End-of-Run
M29	Rigid tapping designation
M30	End-of-Run
M98	Subprogram call
M99	Return from the subprogram; the program will be repeatly executed If the code M99 is used for main program ending (namely, the curren program is not called by other programs).
M9000～M9999	Call macro program (Program No. is larger than 9000)

2.1.1 End of Program (M02)

Format: M02

Function: The M02 code is executed in the Auto mode. The automatic run is ended after the other codes of current block are executed; the cursor stops in the block in which the M02 is located and does not return to the head of the program. If the program is to be executed again, the cursor should return to the beginning of the program.

Besides the above-mentioned functions processed by CNC, the functions of code M02 also can be defined by the PLC ladder diagram. The function defined by standard ladder diagram can be: the current input state of CNC is not change after the code M02 is executed.

2.1.2 Rigid Tapping Designation M29

Format：M29

Function：In auto mode, after the execution of M29, the G74, G84 that followed is processed as在自

rigid tapping codes.

2.1.3 End of run (M30)

Format: M30

Function: If M30 command is executed in the Auto mode, the automatic run is ended after the other commands of current block are executed; the system cancels the tool nose radius compensation and the cursor returns to the beginning of the program when the workpieces number is added by one (whether the cursor returns to the head of the program is determined by parameters).

The cursor does not return to the beginning of the program when the BIT4 of parameter No.005 is set to 0; when it is set to 1, the cursor returns to the beginning of the program as soon as the program execution is finished.

Besides the above-mentioned functions processed by CNC, the functions of code M30 also can be defined by the PLC ladder diagram. The function defined by standard ladder diagram can be: turn OFF the M03, M04 or M08 output signal after the M30 command is executed, and meanwhile output M05 signal.

2.1.4 Subprogram Call M98

Format：M98 P○○○○□□□□

The called subprogram No.（0000 ~ 9999）.The leading zero of subprogram can be omitted when the called times are not input; the subprogram No. should be 4 digits when the called times is input;

Called times（1-9999）calling for once, the input can be omitted

Function: In Auto mode, when the M98 is executed, the subprogram specified by P is called after the execution of other codes in the current block. The subprogram can be performed 9999 times at most. M98 cannot be performed in MDI, or an alarm will occur.

2.1.5 Return from Subprogram (M99)

Format： M99 P○○○○

The block No. (0000～9999) when return to main program is executed, the leading zero can be omitted.

Function: (in subprogram) as the other commands of current block are executed, the block specified by P is performed continuously when the main program is returned. The next block is performed continuously by calling current subprogram of M98 command when returning to the main program; because of the P is not given. If the main program is ended by using the M99 (namely, the current program is not called by other programs for execution), the current program will be run circularly. So, the M99 command is disabled in MDI.

Example: Fig. 2-1shows that the execution route of the subprogram is called (the P command within M99). Fig. 2-2 shows that the execution route of the subprogram is called (the P command is not in M99.

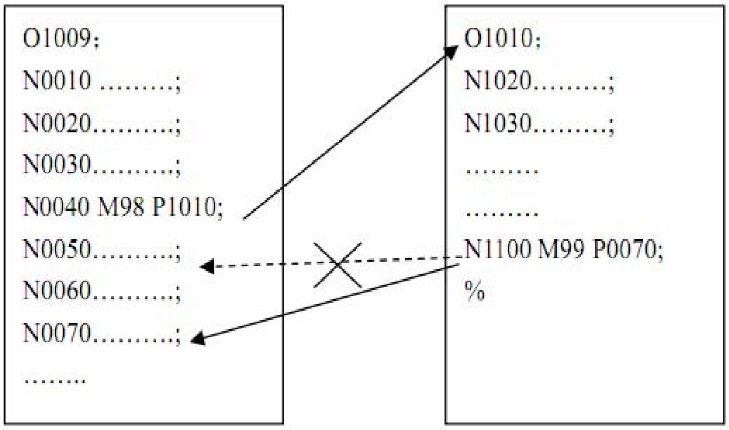
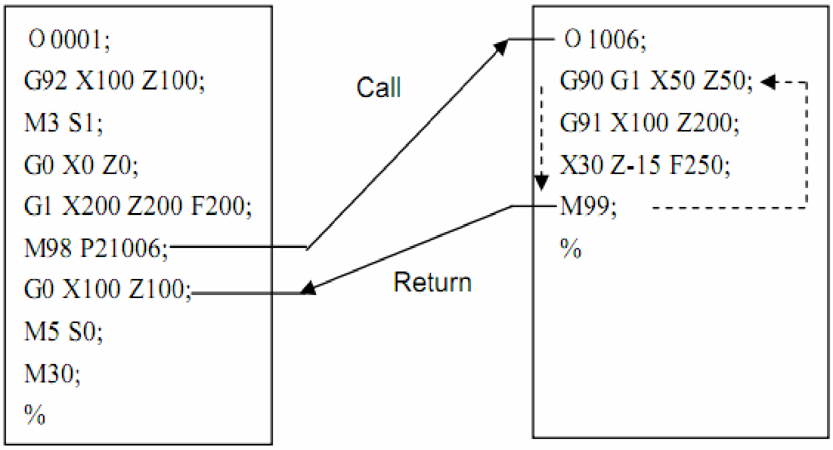


Fig. 2-1



Mainprogram

Fig. 2-2

Subprogram

This GSK980MDa can calls quadruple subprogram, namely, the other subprogram can be called from the subprogram. (See Fig. 2-3)

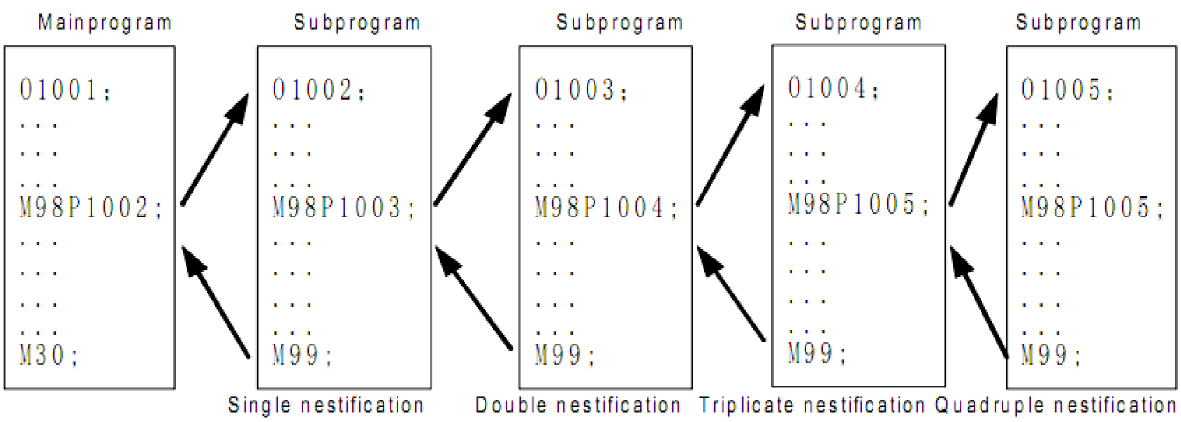


Fig. 2-3 Subprogram nestifications

2.1.6 Macro program call (M9000~M9999)

Format ： M□□□□

9000～9999

Command function: Call the macro program which is corresponded by the command value （O9000～O9999）.

Macro program: Program 09000~09999 is special space obligated for the machine tool manufacturer for using editing and achieving special function subprogram, which is called macro program. Two-level operation authority is needed when editing the program 09000~09999, the user can not modify or run the macro program but the macro calling command if his authority is 3~5 level. So the M9000~M9999 commands are invalid in MDI mode.

2.1.7 M command defined by standard PLC ladder diagram

The M commands other than the abovementioned commands (M02, M30, M98, M99, M9000~M9999) are defined by PLC. The M commands are defined by standard PLC hereinafter. This GSK980MDa milling machine is used for machine control. About the function, meaning, control time sequence and logic etc. of the M command, refer to the manual issued by the machine tool builder.

M command specified by standard PLC ladder diagram

Command	Function	Remark
M00	Program pause	
M03	Spindle CCW	Function interlock, state hold
M04	Spindle CW	
*M05	Spindle stop	
M08	Cooling on	Function interlock, state hold
*M09	Cooling off	
M32	Lubricating on	Function interlock, state hold
*M33	Lubricating off	

Note: The command with " * " specified by standard PLC is valid when the power is on.

2.1.8 Program stop M00

Format: M00

Command function: the program is stopped after executing the M00 command, the "pause" is displayed; the program will continue when the key of Cycle Start is pressed.

2.1.9 Spindle CCW, CW, stop control(M03, M04 and M05)

Format: M03;
M04;
M05;

Command function: M03: spindle forward rotation (CCW);
M04: spindle reverse rotation (CW);
M05: spindle stop.

Note: The control time sequence and logic of M03, M04 and M05 are specified by standard PLC program, refer to the Appendix of this manual.

2.1.10 Cooling control (M08, M09)

Format: M08;
M09;

Command function: M08: cooling on;
M09: cooling off.

Note: The control time sequence and logic of M08 and M09 are specified by standard PLC
program, refer to the Appendix of this manual.

2.1.11 Lubricating control (M32,M33)

Format: M32;
M33;

Command function: M32:lubricating on; M33: lubricating off.

Note: The control time sequence and logic of M32 and M33 are specified by standard PLC program, refer to the Appendix of this manual.

2.2 Spindle Function

The spindle speed is controlled by S command, there are two ways to control spindle speed for GSK980MDa.

Spindle speed switching value control mode: the S□□ (2-digit command value) command is processed by PLC program for exporting the switching value signal to machine, so that the step speed change of the spindle is achieved.

Spindle speed analog voltage control mode: the actual spindle speed is specified by the S□□□□ (4-digit command value), the NC outputs the 0~10V analog voltage signal to the spindle servo device or inverter for achieving the stepless speed regulating of the spindle.

2.2.1 Spindle Speed Switch Value Control

The spindle speed is on switching value control when the BIT4 of bit parameter NO.001 is set to 0. One block only has one S command. The CNC alarm occurs when there are two or more S commands displayed in block.

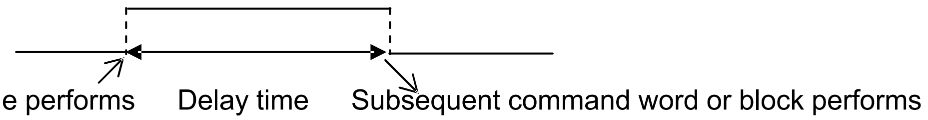
When the S command shares the same block with the command word, the performance sequence is defined by PLC program. For details, refer to the manual issued by the machine tool builder.

This GSK980MDa milling machine is used for machining control when the spindle speed switching value is controlled. The time sequence and logic for S command should be referred by the manual issued by the machine tool builder. The following S command is defined by GSK980MDa standard PLC, for reference only.

Command format： S□□

└── 00～04 (the leading zero can be omitted): 1～4 gears spindle speed switching value control.

In spindle speed switching value control mode, the FIN signal is returned after the set time is delayed after the code signal of S command is sent to PLC. Now the time is called execution time of S code.



The S01, S02, S03 and S04 output states are invariable when the CNC is reset.

The S1~S4 commands are ineffective output when the CNC is switched on. An arbitrary command is performed from S01, S02, S03 and S04, the corresponding S signal output is effective and held on, at the same time the other 3 S signal output are cancelled. The S1~S4 output are cancelled when performing the S00 command, only one of S1~S4 is effective in the meantime.

2.2.2 Spindle speed analog voltage control

The spindle speed is analog voltage control when the BIT4 of current bit parameter is set to 1

Format：S OOOO

└── 0000～9999 (leading zero can be omitted): Spindle speed analog voltage control

Command function: The CNC outputs 0~10V analog voltage to control the spindle servo or inverter for achieving the stepless speed regulating of the spindle when the spindle speed is set. The S command value is not memorized when the power is turned off; and then the parameter recovers to 0 when the power is turned on.

The CNC owns four mechanical spindle shifts function. Counting the corresponding analog voltage value specified by the speed based upon the current set value (corresponding to data parameter No.101~No.104) of the top speed (output analog voltage is 10V) of the spindle shift when the S command is performed, then output the voltage value to spindle servo or inverter, so that the consistency of actual speed and required speed of the spindle are controlled.

The analog voltage output is 0V when the CNC is switched on. The output analog voltage value is invariable (Unless the cutting feed in constant linear speed control and the absolute value of X axis absolute coordinate value are changed) after the S command is executed. The analog voltage output is 0V when the command S0 is executed. And the analog voltage output value is invariable when the CNC is reset or at emergent stop.

The parameter related to spindle speed analog voltage control:

Data parameter No.099: the output voltage offset for spindle top speed (the output analog voltage is 0V); Data parameter No.100: the voltage offset for the zero spindle speed (the output analog voltage is 10V);

Data parameter No.101~No.104: The top speed for spindle 1~4 shifts (the output analog voltage is 10V);

2.2.3 Spindle override

The spindle actual speed can be modified by using spindle override when the spindle speed analog voltage control is effective, the actual speed modified by spindle override is limited by the top speed of current spindle shift, and also it is controlled by the lowest spindle limitation value and the top spindle limitation value in constant linear speed control mode.

This NC offers 8-level spindle override (50%~120%, the change is 10% per level). The actual level and the modificative mode of the spindle override are defined by PLC ladder diagram. Refer to the manual issued by the machine tool builder when attempting to use it. The following description is GSK980MDa standard PLC ladder diagram function, for reference only.

The spindle override defined by GSK980MDa standard PLC ladder diagram has 8 levels. The spindle actual real-time speed can be adjusted by using the spindle override key in the command speed range of 50%~120%, the spindle override will be memorized when the power is turned off. Refer to the OPERATION of this manual for modification operation of the spindle override.

2.3 Tool Function

There is no tool function in this CNC system.

2.4 Feeding Function

2.4.1 Cutting feed (G94/G95, F command)

Format: G94F_; (F0001~F8000, leading zero can be omitted, for feedrate per minute, mm/min)

Command function: The cutting feedrate is specified by mm/min, G94 is modal G command. If the current mode is G94 that it needs no G94 any more.

Format: G95F_; (F0.0001~F500, leading zero can be omitted)

Command function: The cutting feedrate is offered by the unit of mm/rev., G95 is modal G command. The G95 command can be omitted if the current mode is G95. When the CNC performs G95 F_, the cutting feedrate is controlled by feedrate command based on the multiplication of F command value (mm/rev) and current spindle speed (rev/min). The actual feedrate varies with the spindle speed. The spindle cutting feedrate per revolution is specified by G95 F_, the even cutting line can be formed on the face of workpiece. It is necessary to install spindle encoder when the G95 mode is operated.

The G94 and G95 are modal G commands at the same group, one of them is available only.

The G94 is initial state G command, so, it defaults the G94 when the CNC is switched on. The following below shows the conversion formula of feed value per rev. and feed value per min:

$$F_m = F_r \times S$$

There into: F_m: feed value per minute (mm/min);

F_x: feed value per revolution (mm/r);

S: spindle speed (r/min).

The feedrate value is set by the CNC Data parameter No.172 when the CNC is switched on, the F value is invariable after the F command is executed. The feedrate is 0 after F0 is executed. The F value is invariable when CNC is reset or at emergent stop.

Note: In G95 mode, the cutting feedrate will be uneven when the spindle speed is less than 1 rev./min. The following error will exist in the actual feedrate when the spindle speed vibration occurs.

To guarantee the machine quality, it is recommended that the spindle speed selected in machining is not less than the lowest speed of available torque exported by spindle servo or inverter.

Cutting feed: The CNC makes tool movement path and the path (linear or circular arc) defined by command into consistency (The circular interpolation can be performed by two axis in selected plane when it is circular arc, the helical interpolation is formed by the third axis linear interpolation linkage), by which, the CNC controls three directions movement for X axis, Y axis, Z axis ,4th axis and 5th axis at the same time. The instantaneous speed of movement path in a tangential direction is consistent with the F command value, so this is called CUTTING FEED or INTERPOLATION. The cutting feedrate is supplied by F command, which it is disassembled to each interpolation axis according to the programming path when the CNC performs the interpolation command (cutting feed).

Linear interpolation: The CNC can control the instantaneous speed in the directions of X axis, Y axis , Z axis ,4th axis and 5th axis, so the vector resultant speed in these five directions are equal to the F command value.

$$f_x = \frac{d_x}{\sqrt{d_x^2 + d_y^2 + d_z^2 + d_4^2 + d_5^2}} \bullet F$$

$$f_y = \frac{d_y}{\sqrt{d_x^2 + d_y^2 + d_z^2 + d_4^2 + d_5^2}} \bullet F$$

$$f_z = \frac{d_z}{\sqrt{d_x^2 + d_y^2 + d_z^2 + d_4^2 + d_5^2}} \bullet F$$

$$f_4 = \frac{d_4}{\sqrt{d_x^2 + d_y^2 + d_z^2 + d_4^2 + d_5^2}} \bullet F$$

$$f_5 = \frac{d_5}{\sqrt{d_x^2 + d_y^2 + d_z^2 + d_4^2 + d_5^2}} \bullet F$$

F is vector resultant speed for the instantaneous speed in X, Y and Z axis directions

The d_X is instantaneous increment of the X axis, the f_X is instantaneous speed of X axis.

The d_Y is instantaneous increment of Y axis, the f_Y is instantaneous speed of Y axis.

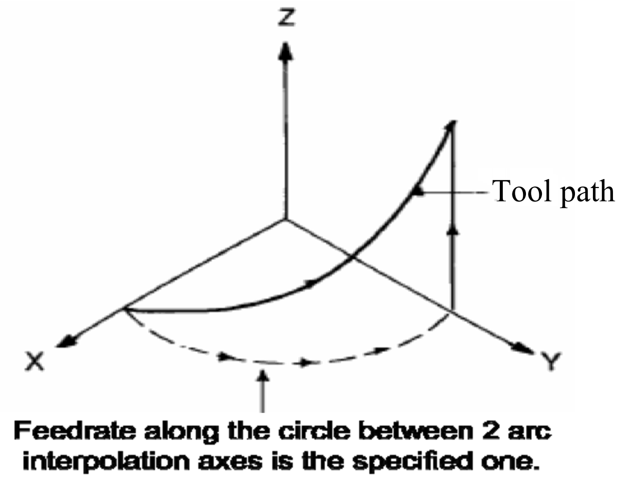
The d_Z is instantaneous increment of Z axis, the f_Z is instantaneous speed of Z axis.

The d_4 is instantaneous increment of 4th axis, the f_4 is instantaneous speed of 4th axis.

The d_5 is instantaneous increment of 5th axis, the f_5 is instantaneous speed of 5th axis.

Circular interpolation (helical interpolation): Performing the arc interpolation in selected plane, the third axis performs linear interpolation, so the F value is circular interpolation speed. An interpolation of linear and circular arc has the following relation when the linear interpolation speed is f:

$$f = F \times \frac{\text{linear axis length}}{\text{circular arc length}}$$



There are 16 levels feedrate override (0~150%, 10% per level) are offered by NC. The actual feedrate series,

the memory performed or not when the power is turned off and the method of overriding are defined by PLC ladder diagram. Refer to the manual issued by the machine tool builder. The function description of GSK980MDa standard PLC ladder diagram is as follows, for reference only.

real-time modification for the cutting feedrate. The actual cutting feedrate can be adjusted in the range of command speed

0~150%, here, the feedrate is memorized when the power is turned off. How to operate the cutting feedrate adjustment, refer to Chapter 3 OPERATION of this manual.

Related parameter:

Data parameter No. 070: the upper limit value (X axis, Y axis, Z axis ,4th axis and 5th axis are same) of the cutting feedrate.

Data parameter No.071: the initial (terminal) speed of exponential acceleration or deceleration for cutting feed.

Data parameter No.072: for exponential acceleration or deceleration time constant of cutting feed.

Data parameter No.073: for initial or terminal speed of exponential acceleration or deceleration in manual feed.

Data parameter No.074: for exponential acceleration or deceleration time constant of manual feed

2.4.2 Manual feed

Manual feed: This GSK980MDa can perform positive/negative movement of X, Y, Z,4th or 5th axis by the current manual feedrate in the Manual mode. X axis, Y axis , Z axis ,4th axis and 5th axis can be moved at one time.

This NC offers 16 levels (0~150%, 10% each time) manual feedrate (override), see the following table 2-2. The actual feedrate series and modification mode or the like in manual feeding, are defined by PLC ladder diagram. Refer to the manual issued by the machine tool builder. The function description of GSK980MDa standard PLC ladder diagram is as follows, for reference only.

Table 2-2

Feedrate override(%)	0	10	20	30	40	0	60	70	80	90	100	110	120	130	140	150
Manual feedrate (mm/min)	0	2.0	3.2	5.0	7.9	12.6	20	32	50	79	126	200	320	500	790	1260

Note: The manual feedrate of X axis is diameter variation per minute; the feedrate defined by GSK980MDa standard PLC ladder diagram is memorized when the power is turned off.

Related parameter:

Data parameter No.073: for speed lower limit of acceleration or deceleration in manual feed.

Data parameter No.074: for exponential acceleration or deceleration time constant in manual feed.

2.4.3 MPG/ Step feed

MPG feed: This GSK980MDa can move positively or negatively in X, Y, Z ,4th or 5th axis by current increment in the MPG mode. Only one of the axis can be moved at one time.

Step feed: This GSK 980MD can move positively or negatively for X, Y, Z ,4th or 5th axis by current increment in the Step mode. One of the axis can be moved only at one time.

Only one mode is effective for the MPG or step mode at one time, it is up to Bit3 of CNC bit parameter No.001.

This NC offers 4 steps (0.001mm, 0.01mm, 0.1mm and 1mm) MPG/step increment. The actual MPG/ step increment series, the selection of increment and current effective axis or the like,

are defined by PLC ladder diagram. Refer to the manual issued by the machine tool builder.

Related parameter: Data parameter No.073: for initial or terminal speed of exponential acceleration or deceleration in manual feed.

Data parameter No.074: for exponential acceleration or deceleration time constant of manual feed.

2.4.4 Automatic acceleration or deceleration

This GSK980MDa performs automatically acceleration or deceleration in order to achieve the smooth transition of the speed at the beginning of the axis movement or before the movement stops; this will diminish the impact when the movement is start or stop. This GSK980MDa adopts kinds of acceleration or deceleration as follows:

Rapid traverse: linear type front acceleration or deceleration Cutting feed: exponential type rear acceleration or deceleration Manual feed: exponential type rear acceleration or deceleration MPG feed: exponential type rear acceleration or deceleration Step feed: exponential type rear acceleration or deceleration

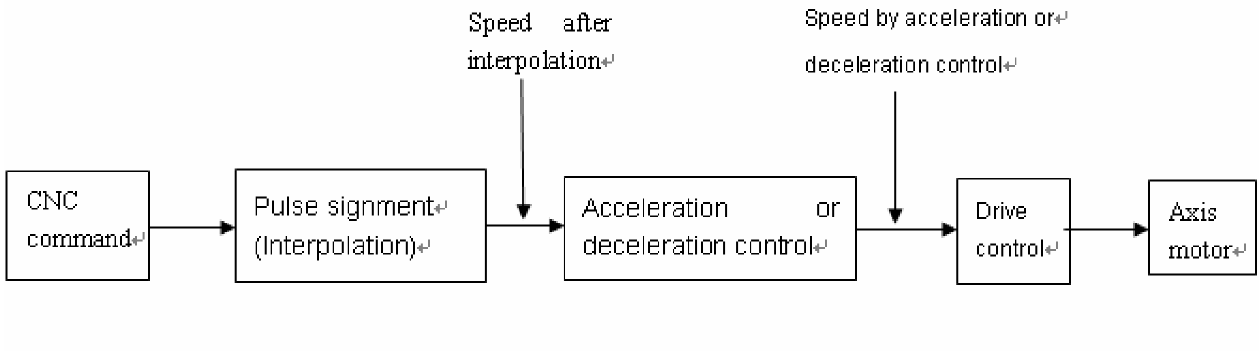
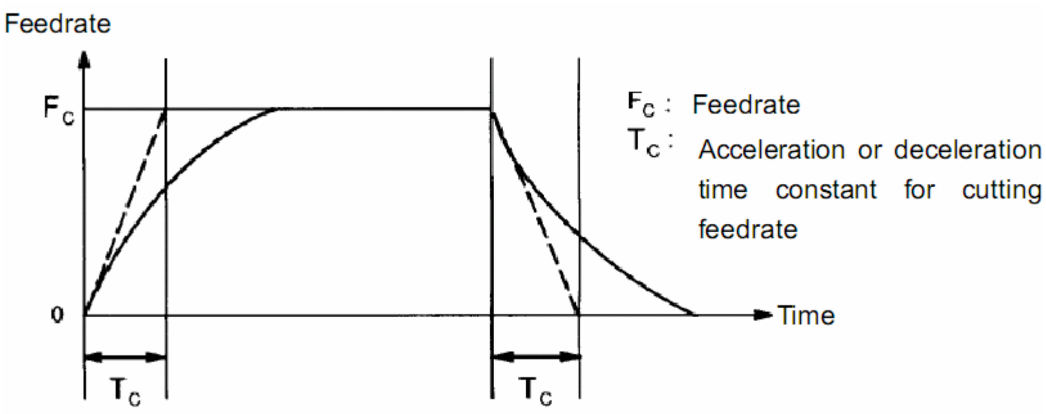


Fig. 2.9



F_C: feedrate

T_C: The acceleration or deceleration time constant of cutting feedrate

(Data parameter No.072 and No.074)

Fig. 2-11 Curves for cutting and manual feedrate

When the cutting feed is performed, this GSK980MDa adopts exponential rear acceleration or

deceleration, an arc transition will be formed for the acceleration or deceleration at the meeting point of the path for the adjacent two cutting feed blocks, when the BIT5 of the bit parameter No.007 is set to 0. A contour error exists between the actual tool path and the programmed path when the positioning is not enough accurate at the meeting point of the two paths.

In order to avoid this kind of error, the exact stop command (G04;) can be inserted between the two blocks or the BIT5 of the CNC bit parameter No.007 is set to 1. Now, the previous block is decelerated to zero speed and it is positioned to the end of the block, and then the next cutting feed block is performed. The following block can be performed because each block is accelerating from the initial speed and then decelerating to zero at last. If the program time is increasing, it may cause the lower machining efficiency.

The SMZ of bit parameter No.007 is set to 0, the transition between two adjacent blocks is processed according to the table 2-3.

Table 2-3

Previous block / Next block	Rapid Position	Cutting feed	Without move
Rapid positioning	X	X	X
Cutting feed	X	O	X
Without move	X	X	X

Note: X: The subsequent block is performed after the previous block is accurately positioned at the end of the block.

O: Each axis speed is transmitted according to the acceleration or deceleration between the adjacent blocks; an arc transition is formed at the meeting point of the tool path.

(Inaccurate positioning)

Example (The BIT3 of the bit parameter is set to 0)

G91 G01*-100; (X axis move negatively)

Z-200; (Z axis move negatively)

Y-300; (Y axis move negatively)

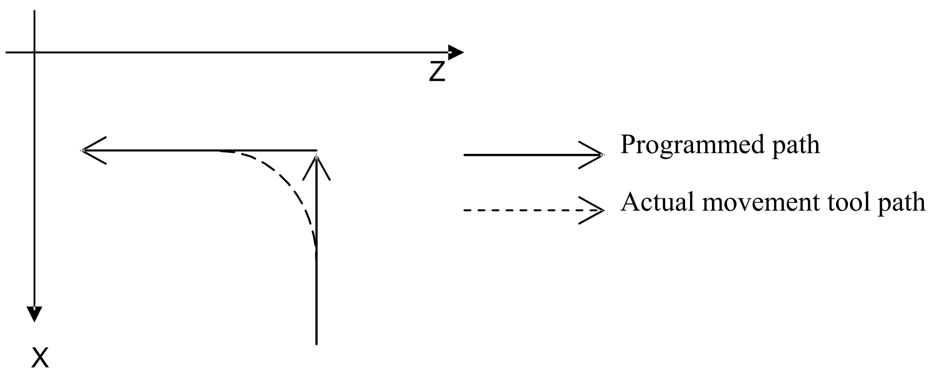


Fig.2-12

CHAPTER3 G COMMAND

3.1 G COMMAND BRIEF

The G command is composed by the command address G and the 1 to 3 digits command value after the command G. Many kinds of operations are specified such as tool movement relative to workpiece, coordinate set, etc. See Table 3-1 for G commands.

G □□□

→ Command value (00~143, the leading zero can be omitted)

→ Command address G

The G command words can be classified into 12 groups such as 00, 01, 02, 03, 05, 06, 07, 08, 09, 10 ,12 and 14. They share the same block except for 01 and 00 groups, different groups G commands can be defined at the same block. The last G command is valid when two or more same group G commands are introduced at the same block. Different G command groups without common parameter (command word) can be defined at the same block, and their functions are simultaneously valid regardless of sequence. If the G command or the optional G command other than Table 3-1 is employed, alarm occurs.

Table 3-1 G command word list

Command word	**Group**	**Function**	**Remark**
G04	00	Dwell, exact stop	Non-modal G command
G28		Machine zero return	
G29		Return from reference point	
G30		2nd, 3rd and 4th reference point return	
G31		Skip function	
G92		Coordinate system set	
G65		Macro	
G00 (initial G command)	01	Rapid traverse	Modal G command
G01		Linear interpolation	
G02		Circular interpolation (CW)	
G03		Circular interpolation (CCW)	
G73		Peck drilling cycle	
G74		Left-hand (counter) tapping cycle	
G80 (initial G command)		Canned cycle cancellation	
G81		Drilling cycle (spot drill cycle)	
G82		Drilling cycle (counter bore cycle)	
G83		Peck drilling cycle	
G84		Tapping cycle	
G85		Boring cycle	
G86		Drilling cycle	
G88		Boring cycle	

G89		Boring cycle	
G110		Circular groove inner rough-milling CW	
G111		Circular groove inner rough-milling CCW	
G112		Circular groove inner fine-milling CW	
G113		Circular groove inner fine-milling CCW	
G114		Excircle finish-milling CW	
G115		Excircle finish-milling CCW	
G134		Rectangle groove rough-milling CW	
G135		Rectangle groove rough-milling CCW	
G136		Rectangle groove inner finish-milling CW	
G137		Rectangle groove inner finish-milling CCW	
G138		Rectangle outer finish-milling CW	
G139		Rectangle outer finish-milling CCW	
G17 (initial G command)	02	XY plane selection	Modal G command
G18		ZX plane selection	
G19		YZ plane selection	
G90 (initial G command)	03	Absolute programming	Modal G command
G91		Relative programming	
G94 (initial G command)	05	Feed per minute	Modal G command
G95		Feed per revolution	
G20	06	Data inch input	Modal power down memorize
G21		Data metric input	
G40 (initial G command)	07	Tool nose radius compensation cancellation	Modal G command
G41		Tool nose radius compensation left	
G42		Tool nose radius compensation right	
G43	08	Tool length offset in + direction	Modal G command
G44		Tool length offset in - direction	
G49 (initial G command)		Tool length offset cancellation	
G140	09	Rectangle path serially punch CW	Non-modal G command
G141		Rectangle path serially punch CCW	
G142		Arc path serially punch CW	
G143		Arc path serially punch CCW	
G98 (initial G command)	10	Return to initial plane in canned cycle	Modal G command
G99		Return to R plane in canned cycle	
G67 (initial G command)	12	Macro program call	Modal G command
G66		Cancel macro program call	
G54 (initial G command)	14	Workpiece coordinate system 1	Modal G
G55		Workpiece coordinate system 2	
G56		Workpiece coordinate system 3	
G57		Workpiece coordinate system 4	

G58		Workpiece coordinate system 5	
G59		Workpiece coordinate system 6	

3.1.1 Modal, non-modal and initial state

The G commands can be set to 12 groups such as 00, 01, 02, 03, 05, 06, 07, 08, 09, 10 ,12 and 14. Thereinto, G commands of 00 group are non-modal G commands, that of other G group are modal commands. G00, G80, G40, G49 ,G67 and G94 are initial G commands.

After the G command is executed, the function defined or status is valid until it is changed by other G command where in the same group, this kind of command is called **modal G command.** After this G command is performed and before the function defined or status is changed, this G command need not be input again when the next block performs this G command.

After the G command is performed, the function defined or status is valid for once, The G command word should be input again while every time the G command is performed, this kind of command is called **non-modal G command.**

The modal G command is valid without performing its function or state after the system is powered on, this is called **initial G command**. If the G command is not introduced after the power is turned on, then the initial G command is executed.The initial commands of GSK980MDa are G00, G80, G40, G49, G67 and G94.

3.1.2 Examples

Example 1

O0001；

G17 G0 X100 Y100;（Move to G17 plane X100 Y100 at the rapid traverse rate; modal command G0 and G17 valid）

X20 Y30; （Move to X20 Y30 at the rapid traverse rate; modal command G0 can be omitted）

G1 X50 Y50 F300; （Linear interpolation to X50 Y50, feedrate is 300mm/min; modal command G1 valid)

X100; （Linear interpolation to X100 Y50, feedrate is 300mm/min; the Y coordinate is not input, use current value Y50; keep F300, the modal command G01 can be omitted）

G0 X0 Y0； （Move to X0 Y0 at the rapid traverse rate, modal G command G0 valid）

M30；

Example 2

O0002；

G0 X50 Y5； （Move to X50 Y5 at the rapid traverse rate）

G04 X4； （Time delay for 4 seconds）

G04 X5； （Time delay again for 5 seconds non-modal command G04 should be

input again)

M30；

Example 3: (the first operation after the power is turned on) O0003；

G90 G94 G01 X100 Y100 F500; （G94 feed per minute，feedrate is 500mm/min）

G91 G95 G01 X10 F0.01; （G95 feed per revolution, input the F value again）

G90 G00 X80 Y50；

M30；

3.1.3 Related definition

The words or characters which are not specially described in this manual are as follows:

Start point: the position before performing the current block;

End point: the position after performing of the current block;

X: the end point absolute coordinate of X axis for G90, the incremental value of X axis against current point for G91;

Y: the absolute coordinate of Y axis at the end for G90, the incremental value of Y axis against current point for G91;

Z: the absolute coordinate of Z axis at the end for G90, the incremental value of Z axis against current point for G91;

F: Cutting feedrate.

3.1.4 Address definition

Usage of the address in system is as follows:

Address	**Function**	**Value range**	**Rounding**
A	Punching number of 1 and 3rd side for rectangle serial punch(G140/G141)	-9999.999~9999.999 Absolute value for negative	Decimal part omitted
	4th，5th axis，axis name address	-9999.999~9999.999	Round-off
B	Punching number of 2nd and 4th side for rectangle serial punch(G140/G141)	-9999.999~9999.999 Absolute value for negative	Decimal part omitted
	Radius for arc serially punch (G142/143)	-9999.999~9999.999	Round-off
	4th，5th axis，axis name address	-9999.999~9999.999	Round-off
C	Punching number for arc serially punch (G142/143)	-9999.999~9999 Absolute value for negative	Decimal part omitted
	4th，5th axis，axis name address	-9999.999~9999.999	Round-off
D	Tool radius offset number	0~32	Decimal

			alarm
E	Unused		
F	G94 feed per minute	0~15000	Decimal efficiency
	G95 feed per rotation	0.0001~500	Round-off
	Tooth pitch in G74,G84 (unit：G21, mm/r;　G20 , inch/r)	0.001~500	Round-off
G	G code	G command in system	Decimal alarm
H	Length offset number	0~32	Decimal alarm
	Operation command in G65	0~99	Decimal alarm
I	Distance from arc start point to center point in X direction	-9999.999~9999.999	Round-off
	G110~G115: radius value of circle	-9999.999~9999.999 Absolute value for negative	Round-off
	G134~G139: width of rectangle in X direction	-9999.999~9999.999 Absolute value for negative	Round-off
	G74,G84：inch screw（unit：tooth/inch）	0.06~25400 Absolute value for negative	Round-off
J	Distance from arc start point to center point in Y direction	-9999.999~9999.999	Round-off
	G112,G113: distance from start point to center point	-9999.999~9999.999 Absolute value for negative	Round-off
	G114,G115: distance from start point to circle	-9999.999~9999.999 Absolute value for negative	Round-off
	G134~G139: width of rectangle in Y direction	-9999.999~9999.999 Absolute value for negative	Round-off
	G140,G141: length of 2nd side of rectangle	-9999.999~9999.999 Absolute value for negative	Round-off
K	Distance from arc start point to the center point in Z direction	-9999.999~9999.999	Round-off
	G110,G111,G134,G135: cutting increment in XY plane each time	-9999.999~9999.999 Absolute value for negative	Round-off

	G136~G139: distance from start point to rectangle side in X axis direction	-9999.999~9999.999 Absolute value for negative	Round-off
L	The length of linear chamfering	-9999.999~9999.999 Absolute value for negative	Round-off
	Punching number for linear serial punch (use together with the canned cycle punch)	-9999.999~9999.999 Absolute value for negative	Decimal part omitted
	Tool life management, tool life value	0~ 999999	Decimal part omitted
M	M miscellaneous function	0~99	Decimal alarm
	M code subprogram call	9000~9999	Decimal alarm
N	Program number	$0\sim2^{31}$	Decimal alarm
	Tool life: tool life unit (0-time, non-0 -time)	0 or other number	Decimal alarm
O	Program number	0~9999	
P	Delay time in G04 (ms)	-9999999~ 9999999 Ignore negative	Decimal alarm
	What kind of number reference return in G30	2~4	Decimal part omitted
	Skip sequence or alarm number in G65	0~9999	Decimal alarm
	M98 subprogram call (times+program name)	0~99999999	Decimal alarm
	Sequence number of M99 subprogram return	0~9999	Decimal alarm
Q	Specifying G73 and G83 cut-in value per time	-9999.999~9999.999 Absolute value for negative	Round-off
	The value of operation in G65	-999999999 ~999999999	Decimal alarm
R	Radius value of arc	-9999.999~9999.999	Round-off
	R plane value of canned cycle command	-9999.999~9999.999	Round-off
	The value of operation in G65	-999999999 ~999999999	Decimal alarm
S	Analog spindle	0~9999	Decimal alarm
	Shift spindle	0~99	Decimal alarm

T	Number of tool	0~32# parameter set value	Decimal alarm
	Tool compensation number	0~32	Decimal alarm
U	Corner radius value of arc corner	-9999.999~9999.999 Absolute value for negative	Round-off
	Corner radius value of rectangle in G134~G139	-9999.999~9999.999 Absolute value for negative	Round-off
V	Distance to unmachined surface, in rapid cut of rough milling command G110,G111,G134 and G135	-9999.999~9999.999 Absolute value for negative	Round-off
W	First cutting-in value in Z direction in rough milling command G110,G111,G134 and G135	-9999.999~9999.999 Absolute value for negative	Round-off
X	Delay time in G04 (s)	-9999.999~9999.999 Absolute value for negative	Round-off
	X axis coordinate value	-9999.999~9999.999	Round-off
Y	Y axis coordinate value	-9999.999~9999.999	Round-off
Z	Z axis coordinate value	-9999.999~9999.999	Round-off

3.2 Rapid PositioningG00

Format: G00 X____ Y____ Z____;

Function: X, Y and Z axes simultaneously move to end points from start at their rapid traverse rates. See Fig. 3-1.

Two axes move at their respective speeds, the short axis arrives at the end firstly, the long axis moves the rest of distance independently, and their resultant paths are possibly not linear.

Explanation: G00, which is initial G command;

The value ranges of X, Y and Z are indicated as -9999.999~+9999.999mm;

X, Y and Z axes, one of them can be omitted or all of them can be omitted. When one of them is omitted, it means that the coordinate value of start and end points are same. The start and end points share the same position when they are omitted at the same time.

Command path figure:

Tool positions at the rapid traverse rate independently for each axis. Usually, the tool path is not linear.

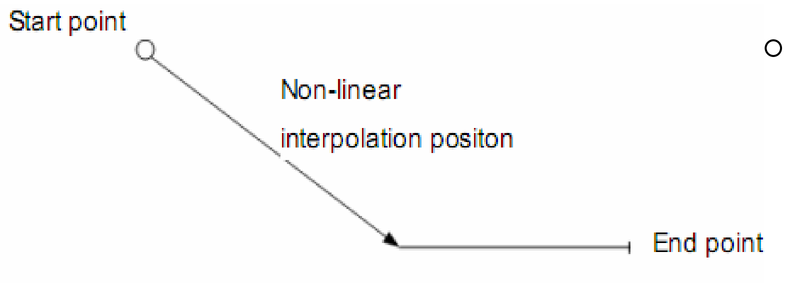


Fig. 3-1

X, Y and Z axes are separately set by the system data parameter No.059, No.060 and No.061 at their rapid traverse rate, the actual traverse rate can be modified by the rapid override keys on the machine panel.

The rapid traverse acceleration or deceleration time constant of X, Y and Z axes are separately set by the system data parameter No.064, No.065 and No.066.

Example: tool traverses from point A to point B. See Fig.3-2.

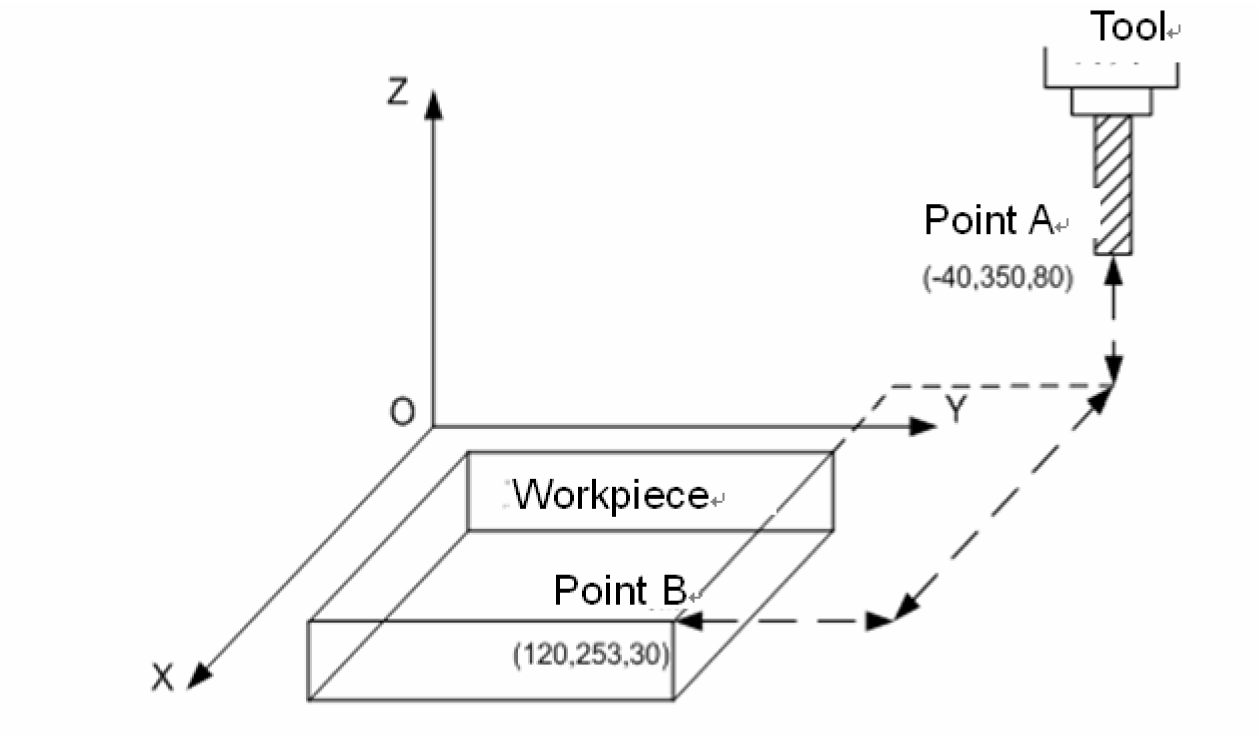


Fig. 3-2

G90 G0 X120 Y253 Z30; (absolute coordinate programming)
G91 G0 X160 Y-97 Z-50; （relative coordinate programming）

3.3 Linear Interpolation G01

Format: G01 X_Y_Z_F_;

Function: Movement path is a straight line from start to end points.

Explanation: G01, which is modal G command;

The value range of X, Y and Z are indicated as -9999.999~+9999.999mm;

X, Y and Z axes which one of them can be omitted or all of them can be omitted.

When one of them

is omitted, it means that the coordinate value of start and end points are consistent. The start and end points share the same position when they are omitted at the same time.

F command value is vector resultant speed of instantaneous rates in X, Y and Z axes directions, the actual feedrate is the product of override and F command value;

F command value is invariable after it is performed till the new one is executed. The following G

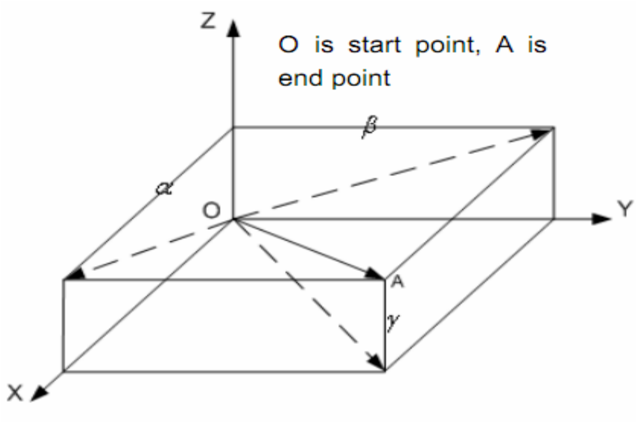
command with F command word uses the same function.

The value range is indicated as follows:

Command function	G94 (mm/min)	G95 (mm/rev)
Value range	1~15000	0.001~500

Command path figure:

The linear interpolation is performed from point O to point A: G01 X α Y β Z γ F f ;



$$L = \sqrt{\alpha^2 + \beta^2 + \gamma^2}$$

The feedrate specified by F is the tool movement speed along the line. The speed of each axis is as follows:

$$\text{Speed in X axis direction: } F_X = \frac{\alpha}{L} \times f$$

$$\text{Speed in Y axis direction: } F_Y = \frac{\beta}{L} \times f$$

$$\text{Speed in Z axis direction: } F_Z = \frac{\gamma}{L} \times f$$

Note: The F initial default value is set by data parameter No.172 when the power is turned on.

3.4 Arc and Helical Interpolation G02, G03

Format：

Circular interpolation:

Arc in the XY plane:

$$G17\left\{\begin{matrix}G02\\G03\end{matrix}\right\} X\_\_\ Y\_\_\ \left\{\begin{matrix}R\_\_\\I\_\_\ J\_\_\end{matrix}\right.\ F\_\_$$

Arc in the XZ plane:

$$G18\left\{\begin{matrix}G02\\G03\end{matrix}\right\} X\_\_\ Z\_\_\ \left\{\begin{matrix}R\_\_\\I\_\_\ K\_\_\end{matrix}\right.\ F\_\_$$

Arc in the YZ plane:

$$G19\left\{\begin{matrix}G02\\G03\end{matrix}\right\} Y\_\_\ Z\_\_\ \left\{\begin{matrix}R\_\_\\J\_\_\ K\_\_\end{matrix}\right.\ F\_\_$$

Helical interpolation

Arc interpolation in XY plane, Z axis linear interpolation linkage;

$$G17\left\{\begin{matrix}G02\\G03\end{matrix}\right\} X\_\_\ Y\_\_\ Z\_\_\ \left\{\begin{matrix}R\_\_\\I\_\_\ J\_\_\end{matrix}\right.\ F\_\_$$

Arc interpolation in XZ plane, Y axis linear interpolation linkage;

$$G18\left\{\begin{matrix}G02\\G03\end{matrix}\right\} X\_\_\ Z\_\_\ Y\_\_\ \left\{\begin{matrix}R\_\_\\I\_\_K\_\_\end{matrix}\right.\ F\_\_$$

Arc interpolation in YZ plane, X axis linear interpolation linkage;

$$G19\left\{\begin{matrix}G02\\G03\end{matrix}\right\} Y\_\_\ Z\_\_\ X\_\_\ \left\{\begin{matrix}R\_\_\\J\_\_\ K\_\_\end{matrix}\right.\ F\_\_$$

Function: Only two axes of circular interpolation can be linked for controlling tool movement along with the arc on the selected plane in any time. If the 3rd axis is specified simultaneously in linear interpolation mode, it will be linked by linear interpolation type to constitute helical interpolation. G02 movement path is CW from start to end points. G03 movement path is CCW from start to end points.

.

Explanation:

G02 and G03 are modal G commands;

R is arc radius, the value range are indicated as -9999.999～9999.999mm;

When the circle center is specified by address I, J and K, they are corresponding with the X, Y and Z axes separately.

I is the difference between the center point and the arc start point in the X axis direction, I= center point coordinate X- X coordinate of arc start point; the value range are indicated as -9999.999～9999.999mm;

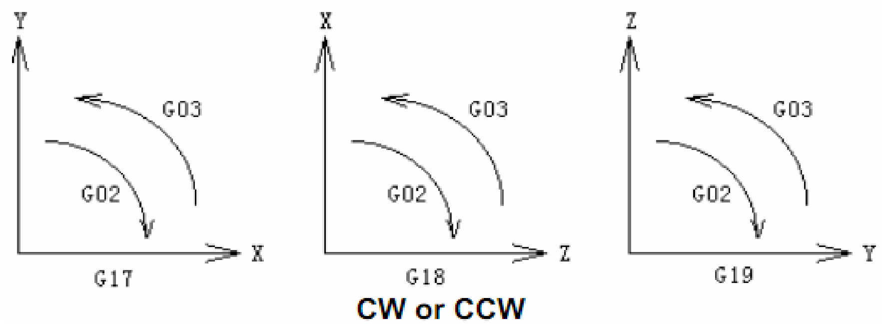
J is the difference between the center point and the arc start point in the Y axis direction, J=center point coordinate Y- Y coordinate of circle arc start point; the value range are indicated as -9999.999～9999.999mm;

K is the difference between the center point and circle start point in the Z axis direction, K=center point coordinate Z- Z coordinate of circle start point; the value range are indicated as -9999.999～9999.999mm.

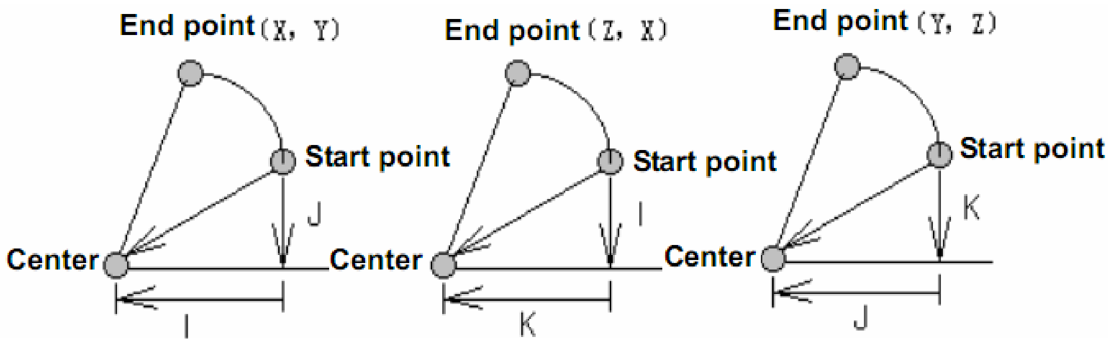
Note When I, J and K are for whole-circle that they have signs according to the direction. And they are positive values when I, J and K share the same directions with X, Y and Z axes; otherwise they are negative ones.

Item	Specified content		Command	Meaning
1	Plane specification		G17	Specifying XY plane arc
			G18	Specifying ZX plane arc
			G19	Specifying YZ plane arc
2	Rotating direction		G02	CW
			G03	CCW
3	End point	G90 mode	Two axes of X, Y and Z	End point in the part coordinate system
		G91 mode	Two axes of X, Y and Z	Distance from start to end points
4	Distance from start point to circle center point		I	X axis distance from start point to the center point (with sign)
			J	Y axis distance from start point to the center point(with sign)
			K	Z axis distance from start point to the center point (with sign)
	Arc radius		R	Arc radius
5	Feedrate		F	Feedrate along the arc

"Clockwise" and "Counterclockwise" are defined when XY plane(ZX plane, YZ plane) is viewed in the positive-to-negative direction of the Z axis (Y axis, X axis) in the Cartesian coordinate system, see the following figure:



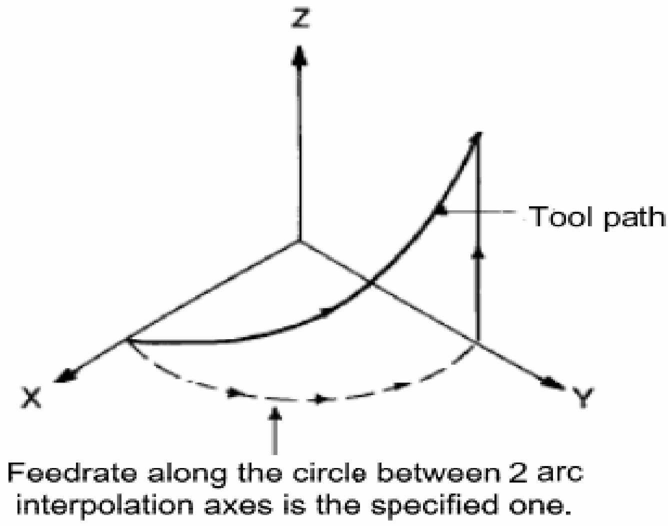
The end point of an arc is specified by using the address X, Y or Z, and is expressed as an absolute or incremental value according to G90 or G91. The incremental value is the distance value from start to end points of an arc. The arc center is specified by address I, J and K against the X, Y and Z respectively. The numerical value following I, J and K, however, is a vector component from start point of an arc to the center point, which is an incremental value with sign. See the following figure:



The F command is circular interpolation rate in helical interpolation, in order to achieve the linkage interpolation between linear axis and arc, the speed of linear interpolation by the 3rd axis has the following relationship to the F command:

$$f = F \times \frac{\text{Length of linear axis}}{\text{Length of circular arc}}$$

Helical interpolation path is as follows:



I, J and K have signs according to the direction. The circular center also can be specified by radius R other than I, J and K, as follows:

G02 X_ Y_ R_ ;
G03 X_ Y_ R_ ;

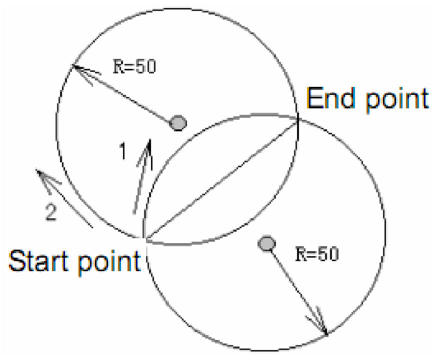
Now, the following two arcs can be described, one arc is more than 180°, the other is less than 180°. The arc radius which is less than 180° is specified by the positive value; the arc radius which is more than 180° is specified by the negative value. The radius is either positive or negative when the arc command is equal to 180°.

(Example) Arc ① less than 180°

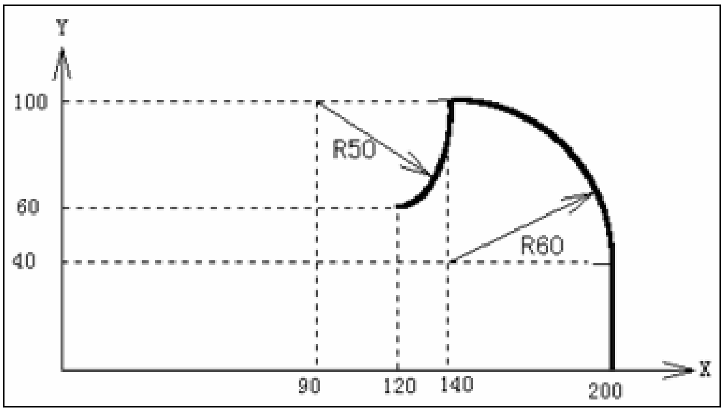
G91 G02 X60.0 Y20.0 R50.0 F300.0;

Arc ② more than 180°

G91 G02 X60.0 Y20.0 R-50.0 F300.0;



(Example for the programming)



To program the above paths using the absolute mode and incremental mode respectively:

(1) Absolute mode

G92 X200.0 Y40.0 Z0 ；

G90 G03 X140.0 Y100.0 I-60.0 F300.0 ；

G02 X120.0 Y60.0 I-50.0 ；

Or G92 X200.0 Y40.0 Z0 ；

G90 G03 X140.0 Y100.0 R60.0 F300.0 ；

G02 X120.0 Y60.0 R50.0 ；

(2) Incremental mode

G91 G03 X-60.0 Y60.0 I-60.0 F300.0 ；

G02 X-20.0 Y-40.0 I-50.0 ；

Or G91 G03 X-60.0 Y60.0 R60.0 F300.0 ；

G02 X-20.0 Y-40.0 R50.0 ；

The feedrate of circular interpolation is specified by F command; it is the speed of the tool along the arc tangent direction.

Note 1:I0, J0 and K0 can be omitted; but, it is very necessary to input one of the addresses I, J, K or R, or the system alarm is generated.

Note 2: The X, Y and Z can be omitted simultaneously when the end and start points share same position. When the center point is specified by address I, J and K, it is a 360° arc.

G02 I_; (Full circle)

The circle is 0° when using R.

G02 R_; (not move)

It is recommended that programming uses R. In order to guarantee the start and end points of the arc are consistent with the specified value, the system will move by counting R again according to the selected plane, when programming using the I, J and K.

Plane selection	**Count the radius R value again**
G17	$R=\sqrt{I^2+J^2}$
G18	$R=\sqrt{I^2+K^2}$
G19	$R=\sqrt{J^2+K^2}$

Note 3: The error between the actual tool feedrate and the specified feedrate is ±2% or less. The command speed is movement speed after tool radius offset along the arc.

Note 4: The R is effective when address I, J and K are commanded with the R, but the I, J and K are disabled at one time.

Note 5: The axis not exists is specified on the set plane, the alarm occurs.

Note 6: If the radius difference between start and end points exceeds the permitted value by parameter (No.100), a P/S alarm occurs.

3.5 Dwell G04

Format: G04 P_ ; or

G04 X_ ;

Function: Axes stop, the current G command mode and the data, status are invariable, after delaying time specified, the next block will be executed.

Explanation: G04, which is a non-modal G-command;

G04 delay time is specified by command words P_, X_;

See the following figure table for time unit of P_ and X_ command value:

Address	**P**	**X**
Unit	0.001 s	s
Available In	0～9999999	0~9999.999

Note:

- **X can be specified by the decimal but P not, or the alarm will be generated.**
- **When the P and X are not introduced or they are negative value, it means exact stop between the**
- **The P is effective when the P and X are in the same block.**
- **The operation is held on when feeding during the G04 execution. Only the delay time execution is finished, can the dwell be done.**

3.6 Plane Selection Command G17, G18 and G19

Format:

G17 ……**XY** plane
G18 ……**ZX** plane
G19 ……**YZ** plane

Function: The plane of arc interpolation and tool radius compensation are chosen by using the G code

Explanation: G17, G18 and G19 are modal G commands, the plane will not be changed when a block without any command inside.

Command example:

G18 X_ Z_ ； ZX plane
X_ Y_ ； invariable plane (ZX plane)

Note:

Note 1: The plane selection command can share the same block with other group G commands.

Note 2: The move command is regardless of the plane selection. For example, the Z axis is not On XY plane, the Z axis movement is regardless of the XY plane in command G17 Z_ .

G17 Z_ ；

3.7 Conversion of Inch and MetricG20 and G21

Format:

G20/G21；

Function: The input unit either inch or metric is chosen by G code.

Explanation:

Unit system	G codes	Min. set unit
Metric	G20	0.0001 inch
Inch	G21	0.001 mm

The G code should be placed in front of the program when inch and metric is switched each other. Before the coordinate system is set, it is specified by a single block command.

The following unit systems vary according to the G code for inch or metric conversion.

(1) Feedrate command value by F.

(2) Command value related to the position.

(3) Offset.

(4) 1 scale value for MPG.

(5) Step amount value.

(6) current coordinate value.

Note 1: The G code for inch or metric conversion when the power is turned on is the same as that at the power off.

Note 2: Changing G20 and G21 are unallowed during programming. Or the alarm occurs.

Note 3: When the unit systems between the machine and input are different, the max. error is 0.5 of the min. move unit; and the error is not be cumulated.

Note 4: As the inch input (G20) and the metric input (G21) switches each other, the offset should be suited to the reset of the input unit.

3.8 Reference Point Return G28

Format: G28 X_ Y_ Z_;

Function: The middle point position specified by X, Y and Z is reached from the start point at the rapid traverse rate, then it returns to the reference point.

Explanation: G28 is a non-modal G-command;

X: The absolute coordinate of middle point in X axis is indicated by G90, the middle point increment against current point in X axis is indicated by G91;

Y: The absolute coordinate of middle point in Y axis is indicated by G90, the middle point increment against current point in Y axis is indicated by G91;

Z: The absolute coordinate of middle point in Z axis is indicated by G90, the middle point increment against current point in Z axis is indicated by G91.

One of the command address X, Y and Z or all of them can be omitted, as follows:

Command	Fun
G28	3 axes hold on at the initial position, the next block continued.
G28 X	X axis reference point return, Y and Z axes still in the original position
G28 Y	Y axis reference point return, X and Z axes still in the original position
G28 Z	Z axis reference point return, X and Y axes still in the original position
G28 X____Z	X and Z axes reference point return simultaneously, Y axis in the origina
G28 X____Y	X and Y axes reference point return simultaneously, Y axis in the origina
G28 Y____Z	Y and Z axes reference point return simultaneously, X axis in the origina
G28 X___Y___Z___	X, Y and Z reference point return simultaneously

Process for command action (See the figure 3-10)**:**

(1) Positioning from current position to intermediate point of command axis at the rapid traverse rate (From point A to B)

(2) Positioning to the reference point from intermediate point at the rapid traverse rate (From point B to R)

(3) If the machine tool is unlocked, the zero return indicator lights up when the reference point return is finished.

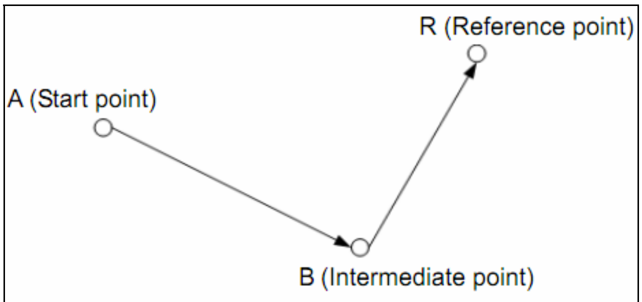


Fig. 3-10

Note:

- After power-on, if G28 is executed prior to the manual machine zero return, the process of G28 machine zero return should be consistent with manual machine zero return, and the deceleration signal and one-rotation signal should be detected. The G28 machine zero return hereafter will not detect the deceleration signal and one-rotation signal, but directly position to zero point.
- During the process of point A→B and B→R, the two axes move at two independent speeds, therefore, the paths may not be linear.
- After the execution of G28 machine zero return, the bit 7 of parameter No.22 decides whether cancel cutter compensation or not.
- In compensation mode, if command G28 is specified, the compensation will be cancelled in the intermediate point. The compensation mode is cancelled automatically after reference point return.
- If zero point switch is not equipped on the machine tool, G28 command and machine zero return are disabled.
- The intermediate point can only be established during the movement from the intermediate point to the reference point which is followed the movement from the start point to the intermediate point.
- After the modification of parameters which set the zero return point, manual reference point return is necessary; G28 command can be executed later.

3.9 Return from Reference Point G29

Format: G29 X_ Y_ Z_;

Function: When a rapid traverse is performed from the current point to mid point, it positions to the specified position by X, Y and Z at the rapid traverse rate.

Explanation:

X: The absolute coordinate of aim point in X axis is indicated by G90; the aim point increment against the mid point in X axis is indicated by G91;

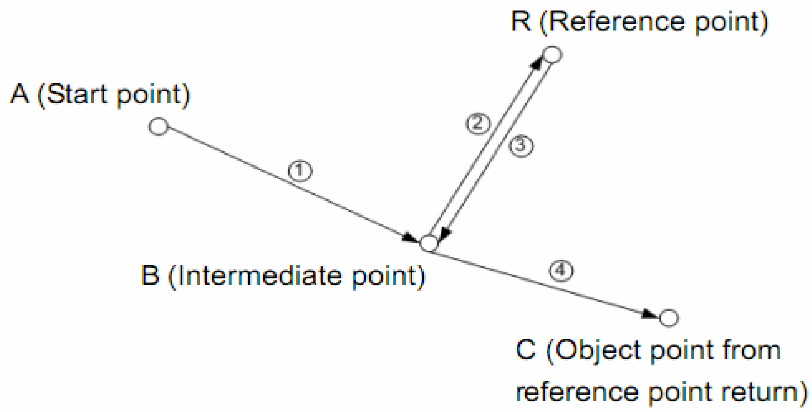
Y: The absolute coordinate of aim point in Y axis is indicated by G90; the aim point increment against the mid point in Y axis is indicated by G91;

Z: The absolute coordinate of aim point in Z axis is indicated by G90; the aim point increment against the mid point in Z axis is indicated by G91;

One of the command address X, Y and Z or all of them can be omitted, see the following figure:

Command	Functi
G29	X,Y and Z axes are in the original position, the next block continued
G29 X	Only X axis performs the command returning from the reference point
G29 Y	Only Y axis performs the command returning from the reference point
G29 Z	Only Z axis performs the command returning from the reference point
G29 X_ Z	Only X and Z axes perform the command returning from the reference point
G29 X__ Y	Only X and Y axes perform the command returning from the reference point
G29 Y___Z	Only Y and Z axes perform the command returning from the reference point
G29 X___Y__Z___	X, Y and Z perform the command returning from the reference point

Process for command action:



(1) The command axis direction performs positioning at the intermediate point specified by G28 (from point R to B), the action is ①→②.

(2) The positioning is performed from intermediate point to specified point (from point B to C), moving to the intermediate and command point at a rapid feedrate, the action is ③→④.

Note:

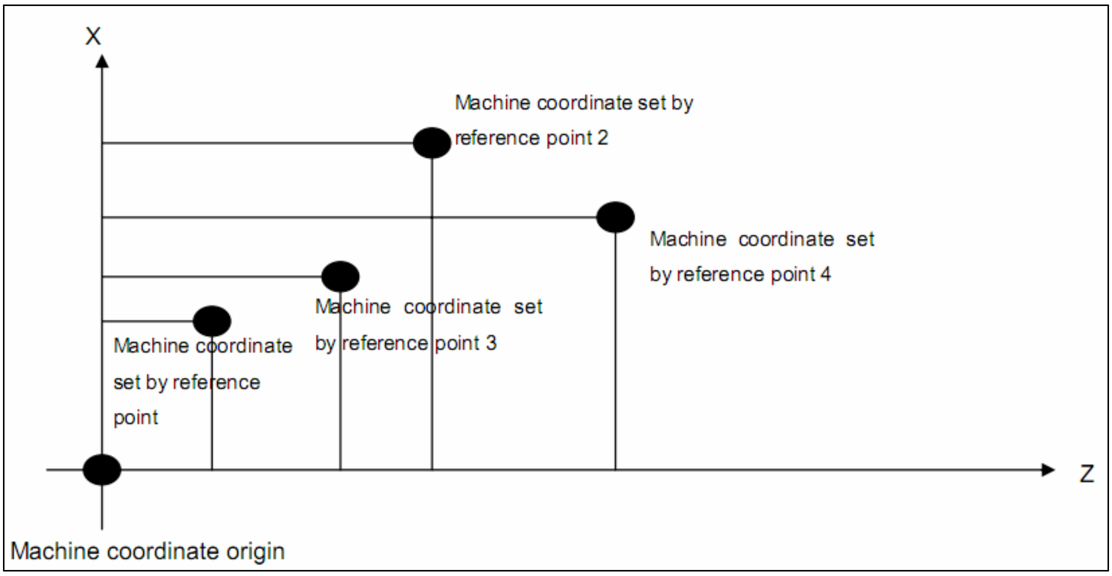
Note 1:G29 is specified after G28, if an intermediate point is not specified by any of axes, the system alarm will be generated.

Note 2: It is incremental distance against the intermediate point in G91 coordinate programming.

Note 3: Current position is reference point when the G29 command is followed to G28 or G30, it returns from reference point directly; or, it returns from current position if G29 command is not followed by G28 or G30.

3.10 The 2nd, 3rd and 4th Reference Point Return G30

Reference point is a fixed point on the machine. By parameters (145#-~164#) it can set four reference points in the machine coordinate system.



Format:

G30 P2 X_ Y_ Z_ ; the machine 2nd reference point return (P2 can be omitted)
G30 P3 X_ Y_ Z_ ; the machine 3rd reference point return
G30 P4 X_ Y_ Z_ ; the machine 4th reference point return

Function: From the start point, after the intermediate point by X, Y and Z is reached at a rapid traverse rate, the machine 2nd, 3rd and 4th reference points are returned. The command word P2 can be omitted when the machine 2nd reference point is returned.

Explanation: G30, which is a non-modal G-command;

X: X axis coordinate for intermediate point;

Y: Y axis coordinate for intermediate point;

Z: Z axis coordinate for intermediate point;

One of the command address X, Y and Z or all of them can be omitted, see the following figure:

Command	**Function**
G30 Pn X	Machine nth reference point return for X axis, Y and Z axes in the original position
G30 Pn Y_ Z_	Machine nth reference point return for Y and Z axes, X axis in the original position
G30	3 axes in the original position, the next block
G30 Pn X_ Y_ Z_	X, Y and Z axes return to the machine nth reference

Note 1:n is 2, 3 or 4 in above table;

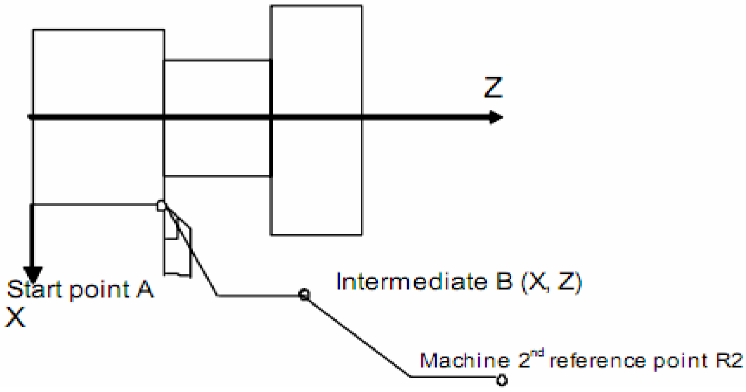
Note 2: Deceleration and zero signals check are not needed when the machine 2nd, 3rd and 4threference points are returned to.

Command action process (see the following figure, an instance of machine 2nd reference point return):

(1) Positioning to intermediate point of the specified axis from current position at a rapid traverse rate (from point A to point B);

(2) Positioning to the 2nd reference position set by data parameter No.94 and No.96 at the setting speed by data parameter No.150 and No.152 (from point B to point R2)

(3) When the reference point returns if the machine is unlocked, the Bit 0 and Bit 1 of the reference point returning end signal ZP21 are HIGH.



Note 1: After returning the machine reference point by manual or the G28 command is performed, the machine 2nd, 3rd and 4th reference point return function can be employed only, or the 2nd, 3rd and 4th reference point operation of G30 command , the system alarm will be generated.

Note 2: From point A to B or from point B to R2, the 2 axes are moved at their separately rate, so the path is not straight line possibly.

Note 3: After machine 2nd, 3rd and 4th reference point returned by the G30 command, the system tool length compensation cancellation is defined by bit 7 of the parameter No.22.

Note 4: The 2nd, 3rd and 4th reference point operation of G30 command can not be executed if the zero switch is not installed on the machine tool.

Note 5: The workpiece coordinate system is set after the machine 2nd, 3rd and 4th reference point are returned.

3.11 Skip Function G31

As G01 linear interpolation is performed, if an external SKIP signal is valid during execution of this command, execution of this command is interrupted and the next block is executed. The skip function is used when the end of machining is not programmed but specified with a signal from the machine, for example, in grinding. It is used also for measuring the dimensions of a workpiece.

Format：

G31 X__ Y__ Z__

Explanation：

1. G31, which is a non-modal G-code, it is effective only in the block in which it is specified.
2. G31 can not be specified in the tool compensation C and chamfering, or the alarm will

be generated. It is very necessary to cancel the tool compensation C and chamfering firstly before the G31 command is specified.

3. Error is allowed in the position of the tool when a skip signal is input.

Signal：The SKIP signal input is on the fixed address X1.0 (XS40-9).

Parameter：

0 1 3							SKPI	G31P

SKIP 1: HIGH level SKIP is valid;

0: LOW level SKIP is valid.

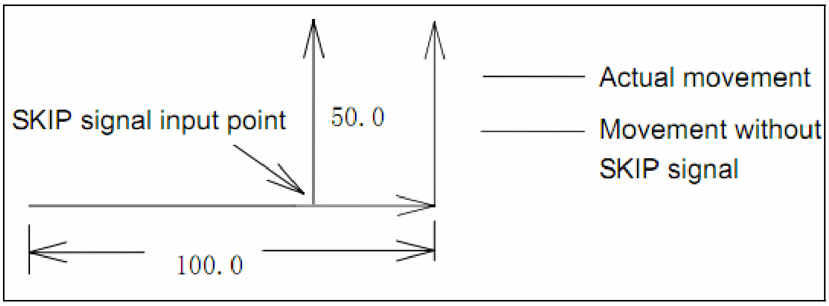
G31P 1: G31 is for immediate stop as the SKIP signal is valid;

0: G31 is for decelerating stop as the SKIP signal is valid.

1. The next block to G31 is incremental command 1: it moves with incremental value from the position interrupted by the skip signal.

Example: G31 G91 X100.0 F100 ；

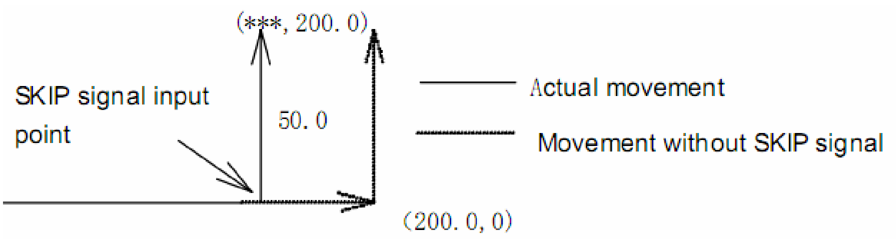
Y50.0 ；



2. The next block to G31 is absolute command for one axis: The command axis moves to the specified position, and the axis not specified keeps at the skip signal input position.

Example: G31 G90 X200.0 F100 ；

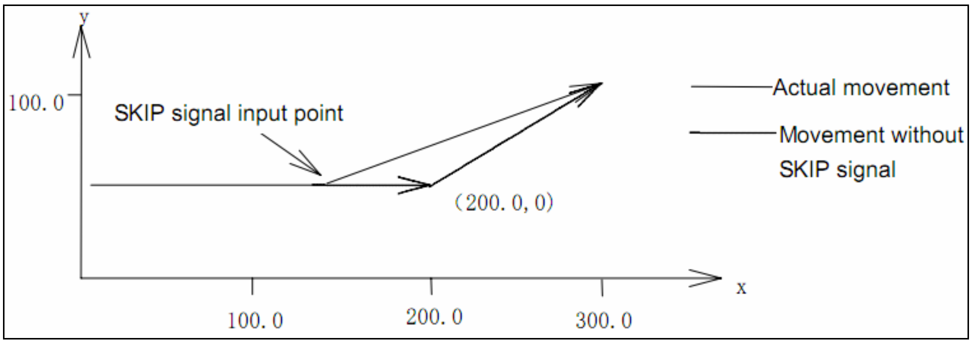
Y100.0 ；



3. The next block to G31 is absolute command for 2 axes: Wherever the skip signal input is, the tool moves to specified position of next block.

Example: G31 G90 X200.0 F100 ；

X300.0 Y100.0 ；



3.12 Tool Nose Radius Compensation C (G40, G41 and G42)

Format:

$$\left.\begin{matrix} G17 \\ G18 \\ G19 \end{matrix}\right\} \left.\begin{matrix} G41 \\ G42 \end{matrix}\right\} D\_\_$$

Functions:

Tool nose radius compensation function

To cancel or perform the tool radius compensation vector by using the commands G40, G41 and G42. They are combined with the commands G00, G01, G02 and G03 for specifying a mode which can be confirmed the compensation vector value, direction and the direction of tool movement.

G codes	Functions
G40	Tool radius compensation cancellation
G41	Tool radius left compensation
G42	Tool radius right compensation

G41 or G42 drives the system into compensation mode; G40 cancels the system compensation mode.

Explanation:

- Compensation plane

The compensation plane can be confirmed based upon plane selection command; the tool compensation C is calculated in this plane.

Plane selection	Plane compensation
G17	X—Y plane
G18	Z—X plane
G19	Y—Z plane

- Compensation value (D code)

This system can be set for 32 compensation values at most. Two digits specified by D code in the program, is called serial number of compensation value, the compensation value should be set by MDI/LCD unit.

D code determines the compensation value in tool offset page according to the bit 1 of parameter No.003, it is very important to notice that the value applied is diameter or radius.

Setting range of compensation value is as follows:

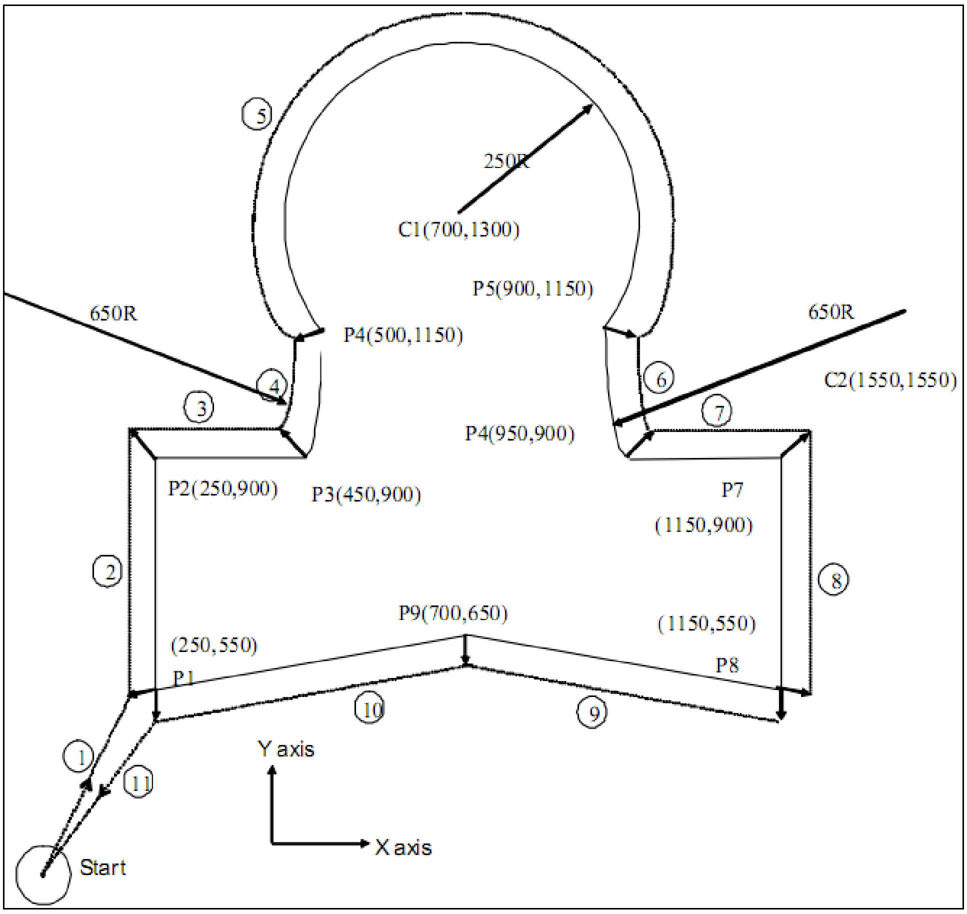
	Millimeter Input（mm）	Inch input（inch）
Compensation value	0～+9999.999mm	0～+999.999 inch

- Compensation vector

The compensation vector is two-dimensional vector; it is equal to the compensation value specified with D code. The compensation vector is calculated in control unit, its direction is real-time modified along with the tool path in each block. You can calculate how much compensation is needed for tool movement when the compensation value is applied in control unit. Compensation path (tool center path) = programmed path . tool radius (or diameter) (determined by compensation direction).

Note:

- Compensation operation is executed in the plane selected by G17, G18, G19. For example, when XY plane is selected, (X,Y) or (I, J) is used to carry out compensation operation and vector operation. The coordinate value whose axis in not in the compensation plane is not affected by the compensation.
- In 3-axis linkage control, compensation only performed for the tool path projected on the compensation plane.
- The alteration of compensation plane should be executed posterior to the compensation mode cancelled. Otherwise, the system will give an alarm and machine stops.
- When the cutter compensation is cancelled by G40, movement amount should be specified, otherwise, an alarm will occur.
- In the canned cycle G codes, G40, G41, G42 codes are disabled.



Example :

Block (1) is named start; the compensation cancellation mode becomes compensation mode by G41 in this block. At the end of this block, tool center is compensated in the direction that tool radius is vertical to next program path (From P1 to P2). Tool compensation value is specified with D07, so set the compensation number to 7, then the G41is indicated with tool path compensation left.

After the compensation begins, tool path compensation performs automatically when creating the workpiece as P1→P2……P8→P9→P1.

N00 G92 X0 Y0 Z0;

N01 G90 G17 G00 G41 D7 X250.0 Y550.0 ； （The compensation value should be pre-set with compensation number）

N02 G01 Y900.0 F150 ；

N03 X450.0 ；

N04 G03 X500.0 Y1150.0 R650.0 ；

N05 G02 X900.0 R-250.0 ；

N06 G03 X950.0 Y900.0 R650.0 ；

N07 G01 X1150.0 ；

N08 Y550.0 ；

N09 X700.0 Y650.0 ；

N10 X250.0 Y550.0 ；

N11 G00 G40 X0 Y0 ；

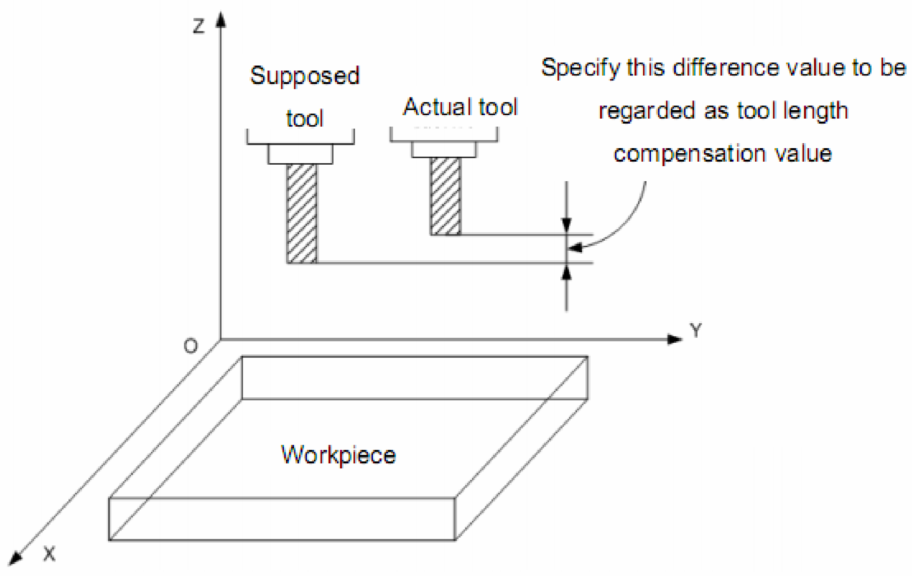
3.13 Tool Length Compensation (G43, G44, G49)

Function: G17 / G18 / G19 } G43 / G44 } H__

Tool length compensation function.

Explanation:

G43 and G44 are modal G codes; they are effective before meeting other G codes in the same group.



The end point specified by Z axis moves an offset value, as above figure G17 plane is selected. Difference between supposed and actual machined tool length value is pre-set at the offset storage when the program is applied. Different length tool can be employed by changing tool length compensation value, so, program change is not needed.

Different offset directions were specified by G43 and G44, the offset number is specified by H code.

Offset axis

The offset axes are vertical to the specified planes (G17, G18 and G19)

Specifying plane	Offset axes
G17	Z axis
G18	Y axis
G19	X axis

Tool position offset for two or more axes can be used to specify the offset axis and the offset axis changed by 2~3 blocks

(Example) X and Y axes compensation

G19 G43 H_ ; …X axis offset

G18 G43 H_ ; …Y axis offset, composed with the previous block, X and Y axes are compensated.

Offset direction

G43: Positive offset

G44: Negative offset

Compensation axes can be regarded as Z, Y and X. Either absolute or incremental command, the end point coordinate value specified by Z axis movement command in program adds the offset specified by H codes in G43 (set in the offset storage), or subtracts the offset specified by H code in G44, finally, the value calculated is regarded as the end point coordinate.

The following command is indicated for Z axis move omitting: When the offset is positive, G43 is for an offset in the positive direction; G44 is for an offset in the negative direction.

It reversely moves when the offset is negative value.

Specifying the offset

An offset number is specified by H code and its corresponding offset adds or subtracts Z axis movement command value in program to get a new Z axis movement command value. The offset number is H00~H32.

Offset value corresponded with offset number is pre-set in the offset storage by using the panel of LCD/MDI. Setting range for offset is as follows:

	Millimeter input（mm）	**Inch input（inch）**
Offset	-9999.999～+9999.999	-999.9999～+999.9999

Offset number 00, i.e. H00 corresponds to the 0 offset. It is disabled to set offset value to H00.

Tool length compensation cancellation

G49 or H00 can be specified when the tool length compensation is cancelled. When two or more axes compensations are cancelled, all of the axes compensation will be cancelled if the G49 is applied. Compensation value of the vertical axis for currently specified plane is cancelled with H00. After G49 or H00 is specified, the system immediately cancels the compensation value.

Note:

1. In the block that tool length compensation is specified, G02，G03, G04, G92 and G31 cannot be specified at the same time, otherwise, an alarm will occur.
2. Tool length compensation command can be specified in the block in which canned cycle is specified. But after the canned cycle is executed, the tool length compensation is disabled and is not modal.

Example：

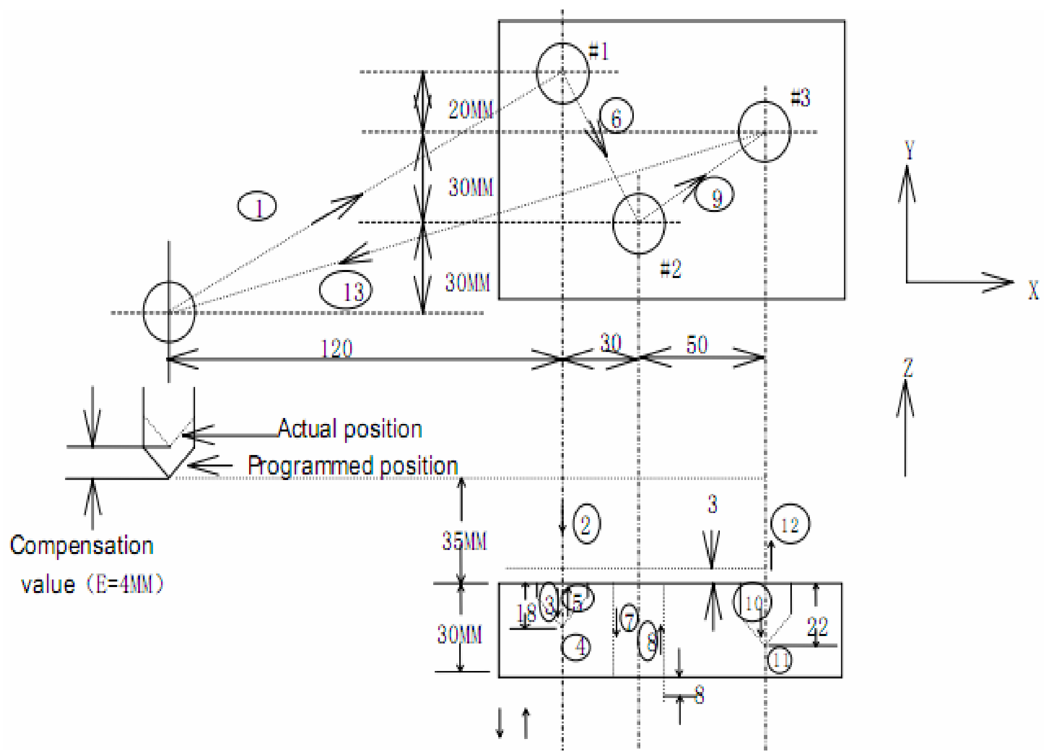
Normal	Modal	Explanation (H1=10.0mm，H2=20.0mm)
G43 H1 G44 G01 X50 Y50 Z50 H2 G90 G00 X100 Y100 Z100	G43 H1 G44 H2 G44 H2	Setting H1, tool length compensation in the positive direction. Linear interpolation, setting H2 tool length compensation in negative direction Position to X100 Y100 Z100(Z80) with H2 compensation offset.

In the same block with G02,G03,G04,G31,G92		
G43 H1 G49 G02 X50 R25 H2	G43 H1 G43 H1	Setting H1 tool length compensation in the positive direction. Alarm occurs.
In the same block with canned cycle code		
G43 H1 G44 G81 X50 R5 Z-70 H2 G90 G00 X100 Y100 Z100	G43 H1 G44 H2 G44 H2	Setting H1 tool length compensation in the positive direction. Setting H2 tool length compensation in the negative direction. Starts the canned cycle from H2.

Specified in the canned cycle		
G43 H1 G90 G81 X50 R5 Z-70 G49 H2 G49 G0 X75 Y75 Z75 H0	G43 H1 G43 H1 G43 H1 G49 H0	Setting H1 tool length compensation in the positive direction. Compensation offset with H1; enters into canned cycle mode. The tool length compensation (G49,H2) in the canned cycle is ineffective, and the previous block remains modal. Cancel all the axis compensations, and set H0 modal. Position to X75 Y75 Z75(Z75).

Command Example:

Tool length compensation (#1, #2 and #3 hole machining)



offset H01 = 4.0

N1 G91 G00 X120.0 Y80.0 ； ……… (1)

N2 G43 Z-32.0 H01 ; ……………… (2)
N3 G01 Z-21.0 ; ……………………… (3)
N4 G04 P2000 ; ………………………(4)
N5 G00 Z21.0 ; ……………………… (5)
N6 X30.0 Y-50.0 ; …………………… (6)
N7 G01 Z-41.0 ; ……………………… (7)
N8 G00 Z41.0 ; ……………………… (8)
N9 X50.0 Y30.0 ; ……………………(9)
N10 G01 Z-25.0 ; …………………… (10)
N11 G04 P2000 ; …………………… (11)
N12 G00 Z57.0 H00 ; …………………(12)
N13 X-200.0 Y-60.0 ; ………………… (13)
N14 M30 ;

Z, X or Y axis offsets a value at offset storage positively or negatively from the original end position according to the above command. Offset axes can be specified with G17, G18 and G19, offset direction can be specified with G43 and G44. Offset No. corresponding to the offset is specified by H code.

3.14 Workpiece Coordinate system G54~G59

Format:

G54 X__ Y__ Z__;	Workpiece coordinate system 1
G55 X__ Y__ Z__;	Workpiece coordinate system 2
G56 X__ Y__ Z__;	Workpiece coordinate system 3
G57 X__ Y__ Z__;	Workpiece coordinate system 4
G58 X__ Y__ Z__;	Workpiece coordinate system 5
G59 X__ Y__ Z__;	Workpiece coordinate system 6

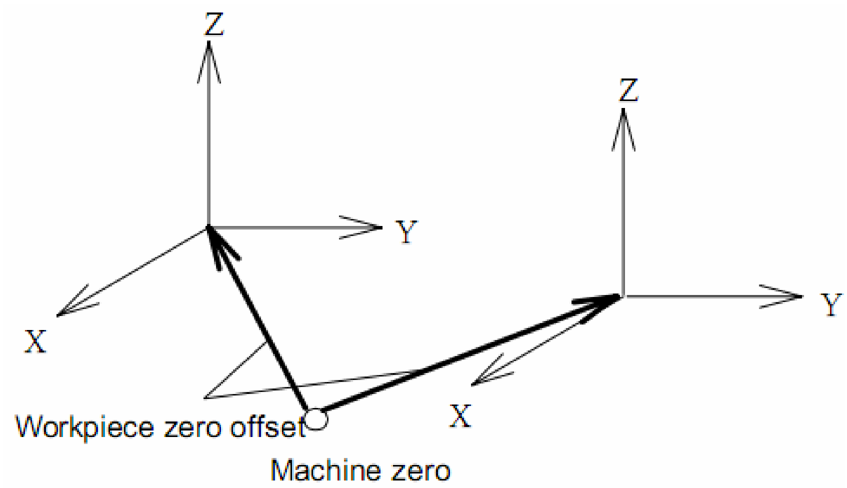
Function:

There are 6 workpiece coordinate systems for machine tool regardless of the G92, any of coordinate system can be selected by G54~G59.

Explanation:

X: New X axis absolute coordinate in current position;
Y: New Y axis absolute coordinate in current position;
Z: New Z axis absolute coordinate in current position.

These six workpiece coordinates are set by the distances (workpiece zero offset) from machine zero to each coordinate system origin.

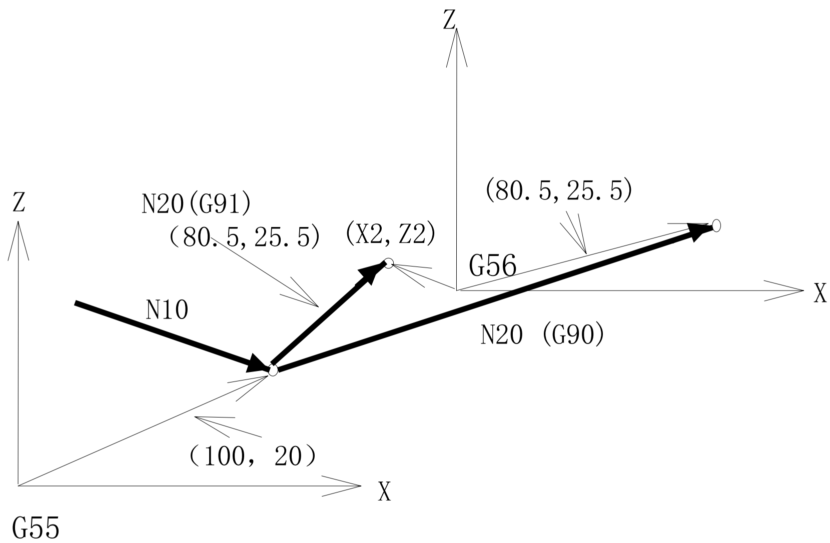


Examples：

N10 G55 G90 G00 X100.0 Z20.0；

N20 G56 X80.5 Z25.5；

Rapidly positioning to workpiece coordinate system 3 (X=80.5, Z=25.5) from workpiece coordinate system 2 (X=100.0, Z=20.0). For example, if N20 block is G91, it is incremental movement. The absolute coordinates automatically become the coordinates in coordinate system G56.



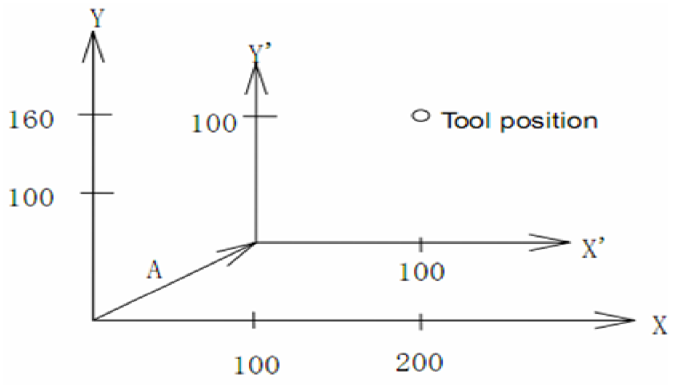
The absolute position for the figure is coordinate value under the current coordinate system.

Note：

- Workpiece coordinate systems 1~6 is set up as soon as machine zero return is executed after power-on. When the system is restarted, the coordinate system is the one set by parameter No. 13 bit 17.
- Whether the relative position varies with coordinate system depends on status parameter №005 PPD. when PPD=0, it changes; when PPD=1, it does not change.
- When the workpiece coordinate system function is determined, usually, G92 is not

needed to set coordinate system. if G92 is used, coordinate system 1~6 will be moved. Do not confuse with G92 and G54~G59, unless workpiece coordinate systems G54~G59 are to be moved. When G54~G59 are in the same block with G92, G54~G59 are disabled.

- Workpiece coordinate system can be modified in the program run. The new coordinate system is effective till the system is restarted.



If it performs G92 X100 Y100 commands when the tool is positioned a（t 200，160）in the G54 coordinate system; the offset vector A for workpiece coordinate system 1 is (X', Y'). And the other workpiece coordinate systems offset for vector A.

3.15 Compound Cycle Command

3.15.1 Brief for canned cycle

Generally, the canned cycle is a machining movement completion from one block with G function to the completion of multi-block specified. Canned cycles make it easier for the programmer to create programs. With a canned cycle, a frequently-used machining operation can be specified in a single block with a G function; without canned cycles, multiple blocks are needed, and canned cycles can shorten the program to save memory.

3.15.1.1 Canned cycle list

G codes	Drilling	Operation at the bottom of a hole	Retraction	Application
G73	Intermittent feed	—	Rapid feed	High-speed peck drilling cycle
G74	Feed	Dwell, spindle CCW	Feed	Left-hand tapping cycle
G80	—	—	—	Canned cycle cancellation
G81	Feed	—	Rapid feed	Drilling, point drilling
G82	Feed	Dwell	Rapid feed	Drilling, boring, counter boring
G83	Intermittent feed	—	Rapid feed	Peck drilling cycle
G84	Feed	Dwell, spindle CW	Feed	Tapping
G85	Feed	—	Feed	Boring
G86	Feed	Spindle stop	Rapid feed	Boring
G88	Feed	Dwell, spindle stop	manual	Boring

G89	Feed	Dwell	Feed	Boring
G110	Intermittent feed	Full-circle helical rough milling	Rapid feed	Round groove internal rough milling CCW
G111	Intermittent feed	Full-circle helical rough milling	Rapid feed	Round groove internal rough milling CW
G112	Feed	Full-circle fine milling	Rapid feed	Full-circle internal fine milling CCW
G113	Feed	Full-circle fine milling	Rapid feed	Full-circle internal fine milling CW
G114	Feed	Full-circle fine milling	Rapid feed	External round fine milling CCW
G115	Feed	Full-circle fine milling	Rapid feed	External round fine milling CW
G134	Intermittent feed	Rectangle rough milling	Rapid feed	Rectangle groove internal rough milling CCW
G135	Intermittent feed	Rectangle rough milling	Rapid feed	Rectangle groove internal rough milling CW
G136	Feed	Rectangle fine milling	Rapid feed	Rectangle groove internal fine milling CCW
G137	Feed	Rectangle fine milling	Rapid feed	Rectangle groove internal fine milling CW
G138	Feed	Rectangle fine milling	Rapid feed	Rectangle groove external fine milling CCW
G139	Feed	Rectangle fine milling	Rapid feed	Rectangle groove external fine milling CW

3.15.1.2 Canned circle explanations

Generally, a canned cycle consists of a sequence of the following operations, see the right figure.

Operation 1… Positioning of axes X and Y

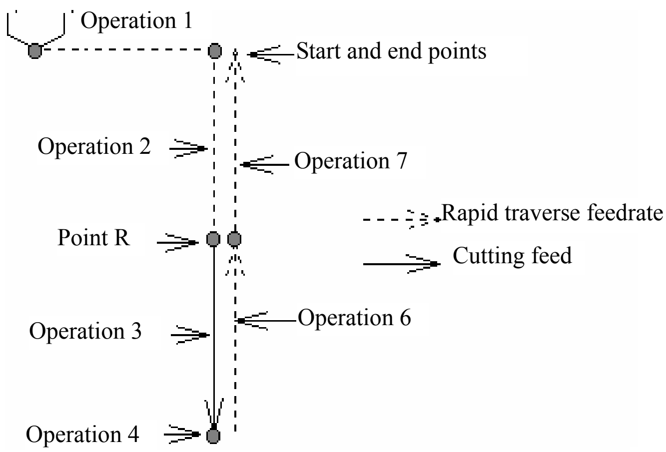
Operation 2…Rapid traverse to point R plane

Operation 3…Hole machining;

Operation 4…Operation at the bottom of hole;

Operation 5…Retraction to point R plane

Operation 6…Rapid traverse to the initial Point



3.15.1.3 G90/G91

The data mode corresponded with G90 and G91 are different. The point R plane and the absolute position machined at the bottom of the hole are specified by R and Z values, when the

command is G 90. The specified R value is the distance relative to the initial plane, and the Z value is the distance relative to the R point plane when the command is G91. See the Fig. 13.1 (B)

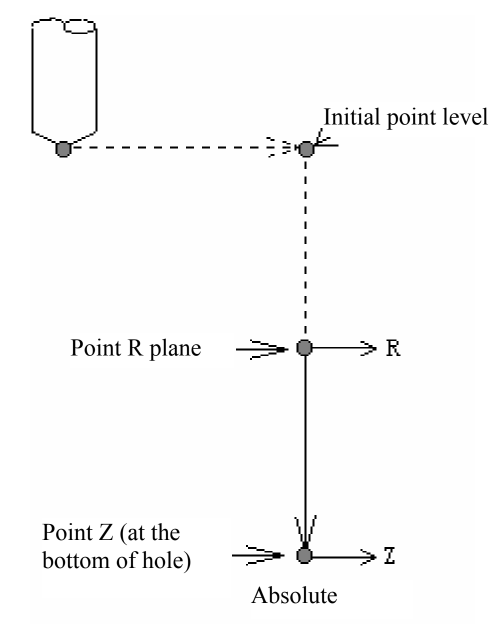
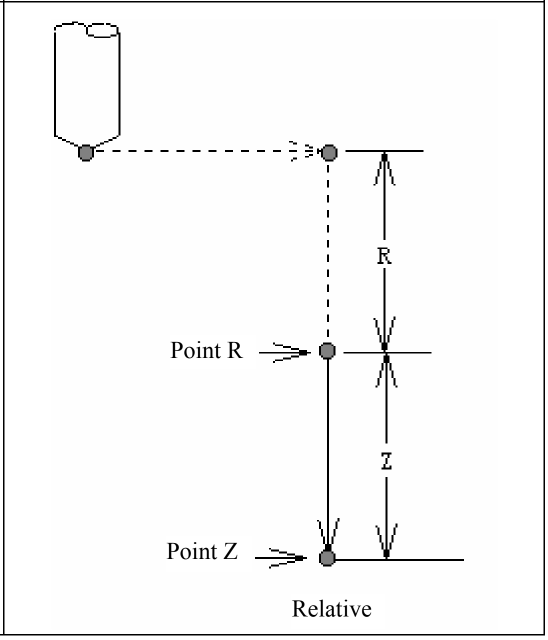
G90 (Absolute command)	G91 (Incremental command)
	

Fig. 13.1 (B) Absolute and incremental commands for canned cycle

3.15.1.4 Returning point level G98/G99

Tool can be returned to the initial plane or point R plane according to G98 and G99 during returning. See the following figure Fig. 13.1 (C).

Normally, the initial hole machining is used by G99, the last machining is used with G98. The initial level will not be changed when the hole machining is done by G99.

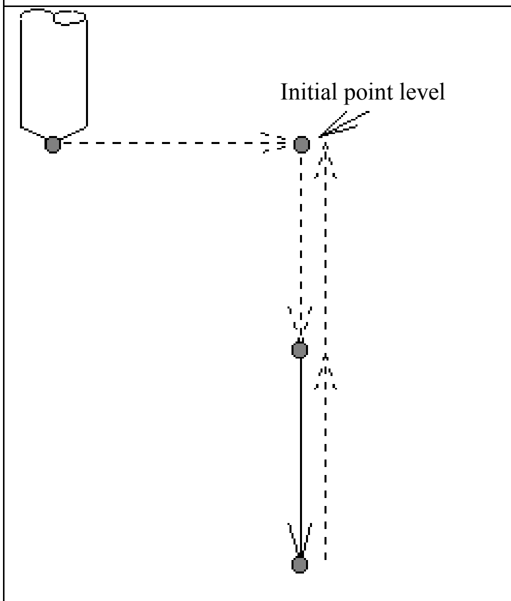
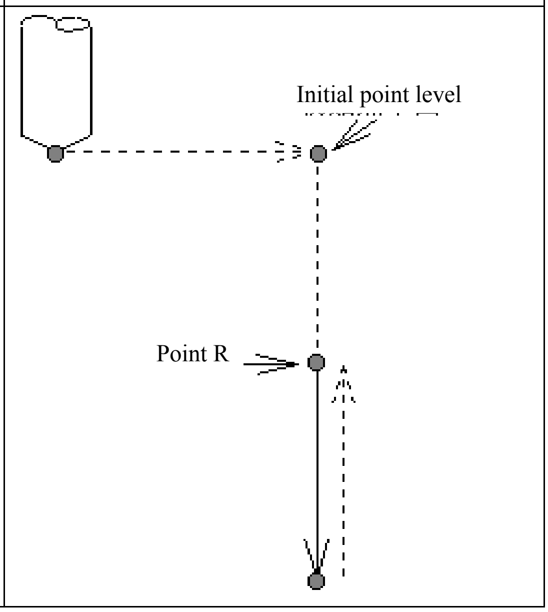
G98 (Return to initial level)	G99 (Return to point R plane)
	

Fig.13.1 (C) Levels for initial and point R

Note ：The initial point level is an absolute position for hole machining axis direction which is indicated from the canned cycle cancellation to start.

3.15.1.5 Canned cycle cancellation

There are two ways for canned cycle cancel are listed below:

1. Canceling the canned cycle with the G80
2. The canned cycle is cancelled by the G00, G01, G02 and G03 command in group 01.

(1) When the canned cycle is cancelled by the command G80, if the G00, G01, G02 and G03 of the 01 group are not specified, then the reserved modal command (G00 or G01) performs motion before using canned cycle.

For example:

N0010 G01 X0 Y0 Z0 F800; （The modal command is G01 before entering the canned cycle）

N0020 G81 X10 Y10 R5 Z-50; （Entering canned cycle）

N0030 G80 X100 Y100 Z100; (The modal G01 command reserved before canned cycle performs cutting feed ）

If the G01 is not specified in the abovementioned program N0010, but G00, the G00 performs rapid positioning for N0030.

When both command G80 and commands G00, G01, G02 and G03 are specified in block, actions are performed by the latter, G00, G01, G02 and G03.

For example:

N0010 G01 X0 Y0 Z0 F800; (The modal command is G01 before entering the canned cycle)

N0020 G81 X10 Y10 R5 Z-50; (Entering canned cycle)

N0030 G00 G80 X100 Y100 Z100; (The G00 performs positioning at the rapid rate, and the modal command G00 is saved)

Note: The cutting feedrate by F command is still held on even if the canned cycle is cancelled.

3.15.1.6 General command format for canned cycle

Once the hole machining data is specified in the canned cycle, it is held until the canned cycle is cancelled. So the hole machining data should be outright specified at the beginning of the canned cycle, only the modified data is specified in the following canned cycle.

The general command format of canned cycle: G_ X_ Y_ R_ Z_ Q_ P_ F_ L;

All commands for canned cycle are listed in above-mentioned format. But it is not needed to specify the above-mentioned format in each canned cycle. For example, the canned cycle can be performed as long as the G command (hole machining) and any of X, Y, Z and R are specified; additionally, Q or P is not available in some canned cycle G command (hole machining), the command is disabled even if these data are specified, they are regarded as modal data memories only.

Table 13.1.7 Command explanations for canned cycle

Specifying content	Address	Explanation for command address
Hole machining	G	Refer to the canned cycle list.
Hole position data	X，Y	Specifying the hole position with the absolute and incremental value, control is same with G00 position. Unit: mm;
Hole machining data	R	See the fig.13.1 (B), the distance from initial point level to point R plane is specified by using the incremental value, or specifying the coordinate value of the point R by absolute value. Unit: mm;
	Z	Hole depth. See the fig.13.1 (A), the distance from R point to the bottom of a hole is specified by using the incremental value or specifying the coordinate value of the hole bottom by absolute value. Unit: mm;
	Q	Specifying each cut-in in G73 and G83 or translational value in G76 and G87. Unit: mm;
	P	Specifying the dwell at the bottom of a hole. Relation of time and the numerical specified are same with G04. Unit: ms;
	L	Machining cycle for L holes is performed from start (start position of block) to XY coordinate position.
	F	The cutting feedrate is specified, tooth pitch is indicated in G74 and G84.

A part of command of canned cycle such as G110, G111, G112, G113, G114, G115, G134, G135, G136, G137, G138 and G139 are explained in the following chapters or sections.

3.15.2 Description for canned cycle

3.15.2.1 High-speed peck drilling cycle G73

Format: G98/G99 G73 X_ Y_ R_ Z_ Q_ F_ L_;

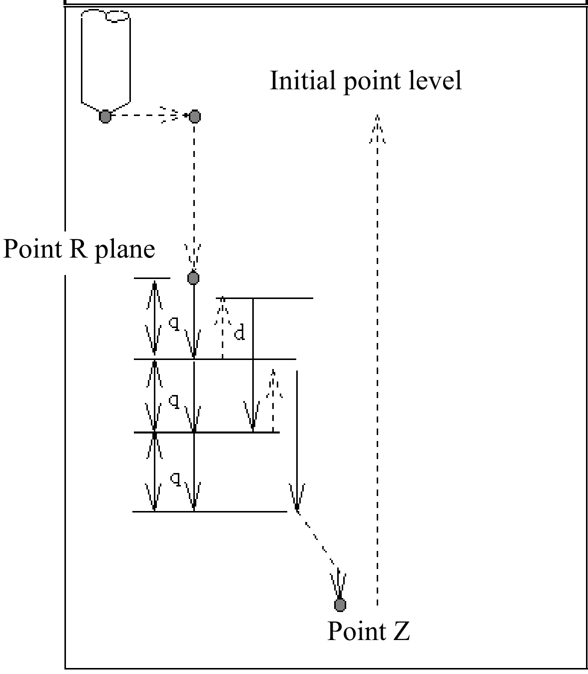
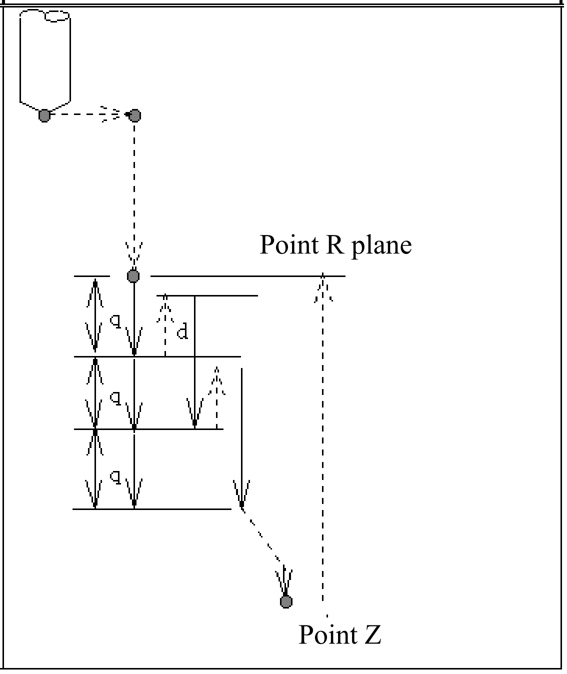
Function: This kind of cycle performs high-speed peck drilling, it performs intermittent cutting feed to the bottom of a hole, and eliminating the chips from the hole simultaneously.

Explanation: Refer to the command explanation of canned cycle in Table 13.1.7.

Cycle process:

(1) Positioning to XY plane level at the rapid traverse;
(2) Down to the point R plane at the rapid traverse rate;
(3) Cutting feed for Q distance;
(4) Retract *d* distance in rapid traverse;
(5) Cutting feed for (Q+d) distance
(6) Machine to the Z axis hole bottom by cycling the (4) and (5);
(7) Return to the start point level or point R plane according to G98 or G99 at the rapid traverse.

Command Path:

G98 Return to the initial plane at the rapid traverse	G99 Return to the point R plane at the rapid traverse
	

Related Explanation:

(1) This kind of cycle is peck drilling for Q value intermittent feeding along the Z-axis direction. The Q value should be positive, the sign is ineffective even if the negative value is specified. If the Q value is not specified, then it defaults 0.1mm. If a depth to be cut is less than the Q value, then cut to the bottom of the hole without tool retraction at the rapid traverse for the first time.

(2) To remove chips from the hole easily, a small value can be set for retraction. This allows drilling to be performed efficiently. The tool is retracted in rapid feed, the retraction amount d is set by parameter No.51, the default is 1000, unit: 0.001mm.

(3) The command P is disabled, but its value is reserved as canned cycle modal value.

3.15.2.2 Left-handed tapping cycle G74

Format: G98/G99 G74 X_ Y_ R_ Z_ P_ F_ L

Function: This cycle performs left-handed tapping. In the left-handed tapping cycle, the spindle rotates clockwise for tapping till the bottom of the hole has been reached, then retracts by counter-clockwise after dwell.

Explanation: For canned cycle explanation, see the Table 13.1.7

Thereinto, the F is indicated for tooth pitch. The value range are indicated as 0.001~500.00mm (metric), 0.06~25400 teeth/inch (inch)

Cycle process:

(1) Positioning to XY plane level at the rapid traverse;

(2) Down to the point R plane at the rapid traverse;

(3) Tapping to the bottom of a hole;

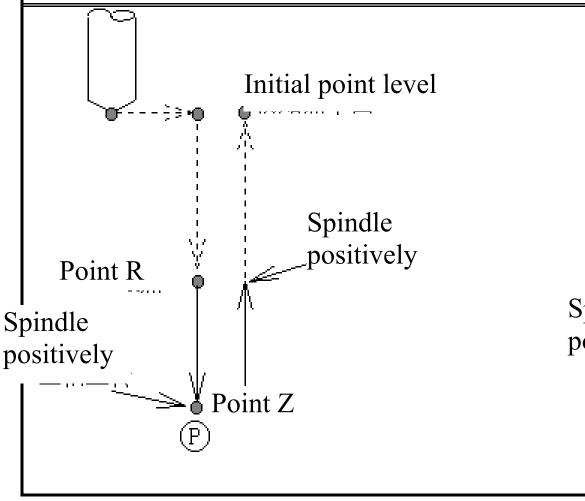
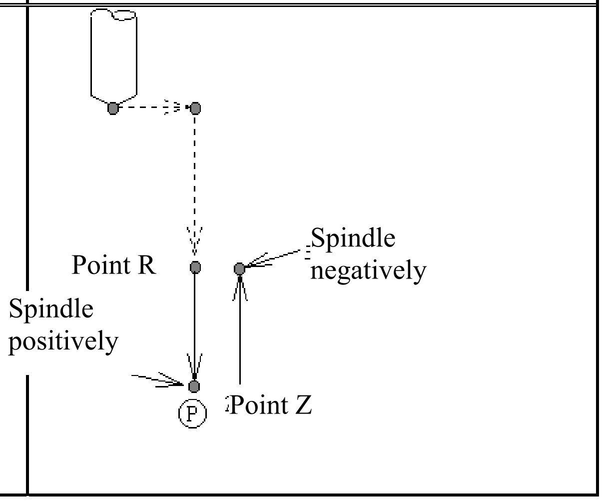
(4) The spindle stops;

(5) Pause for time P if dwell is specified;

(6) The spindle rotates CCW, and then retracts to point R plane;

(7) The spindle is stopped; pause for time P if dwell is specified;
(8) Spindle rotates CW;
(9) Return to the initial plane if it is G98.

Command Path:

G98 (Mode for returning to initial plane)	G99 (Mode for returning to R point plane)
	

Related Explanation:

(1) Tapping to the bottom of a hole it will not be returned immediately even if the P is omitted or regarded as 0 in this cycle, it will be returned after a dwell time (2s), and this time is set by system.

(2) The F is tapping modal value, the last tapping F value is taken when it is omitted, or alarm will be generated if it does not exist.

(3) The metric or inch of the F value is determined by G20 (metric) or G21 (inch).

(4) The command Q is disabled in this cycle, but its value will be reserved as canned cycle modal value.

3.15.2.3 Tapping cycle G84

Format: G98/G99 G84 X_ Y_ R_ Z_ P_ F_ L_ ;

Function: This cycle is used to machine a thread. The tapping is performed by spindle rotating positively, when the bottom of a hole has been reached, the spindle is retracted in the reverse direction.

Explanation: For command explanation of canned cycle, see the Table 13.1.7

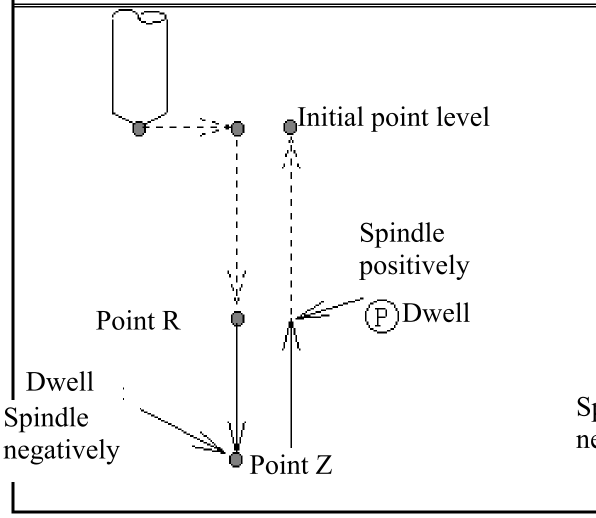
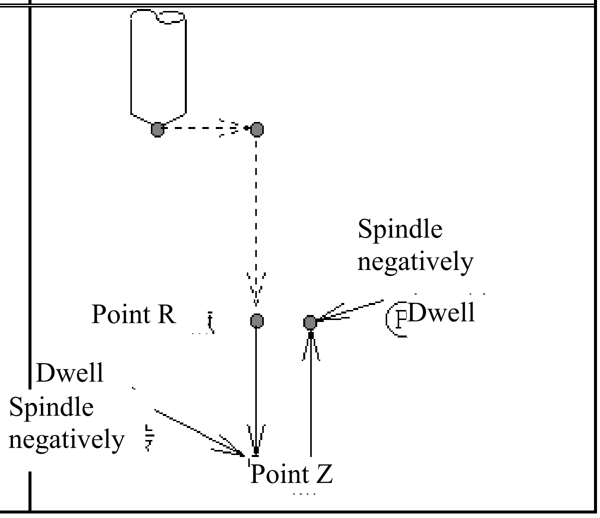
Thereinto, the F is tooth-pitch. The value range is 0.001~500.00mm (metric), 0.06~25400 tooth/inch (inch).

Cycle Process:

(1) Positioning to the XY plane level at the rapid traverse;
(2) Down to the point R plane at the rapid traverse;
(3) Tapping to the bottom of a hole;
(4) Spindle stops;
(5) For dwell time P if it is commanded
(6) Spindle returns to the point R plane in reverse direction;

(7) Spindle stops; for dwell time P if the P is commanded;
(8) The spindle is rotated in the positive direction;
(9) Returning to the initial point level if it is G98.

Command Path:

G98 (Mode for returning to initial point level)	G99 (Mode for returning to point R plane)
	

Related Explanation:

Please refer to the related explanation for G74 (Counter tapping cycle)

3.15.2.4 Drilling cycle, spot drilling cycle G81

Format: G98/G99 G81 X- Y_ R_ Z_ F_ L_ ;

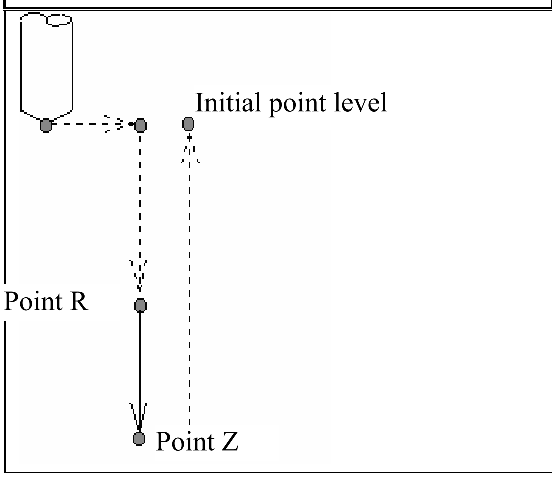
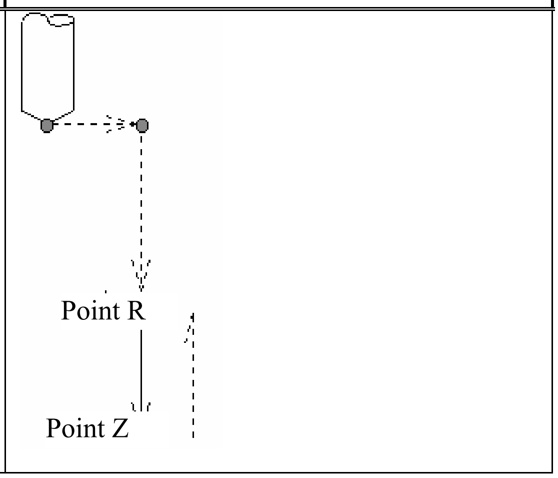
Function: This cycle is used for normal drilling. Cutting feed is performed to the bottom of the hole, the tool is then retracted from the bottom of the hole in rapid traverse.

Explanation: For the command explanation of canned cycle, see the Table 13.1.7.

Cycle Process:

(1) Positioning to the XY plane level position at the rapid traverse;
(2) Down to the point R plane at the rapid traverse;
(3) Cutting feed to the bottom of the hole;
(4) Returning to the initial point or point R plane at rapid traverse according to the G98 or G99;

Command Path:

G98 Return to the initial plane at the rapid traverse	G99 Return to the R point plane at the rapid traverse
	

Related Explanation:

The command Q or P is disabled in this cycle, but its value will be saved as canned cycle modal value.

3.15.2.5 Drilling cycle, counter boring cycle G82

Format：G98/G99 G82 X_ Y_ R_ Z_ P_ F_ L_ ;

Function: Cutting feed is performed to the bottom of the hole. Hole depth precision is added when the dwell is performed, and then the tool is retracted from the bottom of the hole at rapid traverse.

Explanation: For the command explanation of these canned cycles, see the Table 13.1.7

Cycle process:

(1) Positioning to the XY plane level at the rapid traverse;

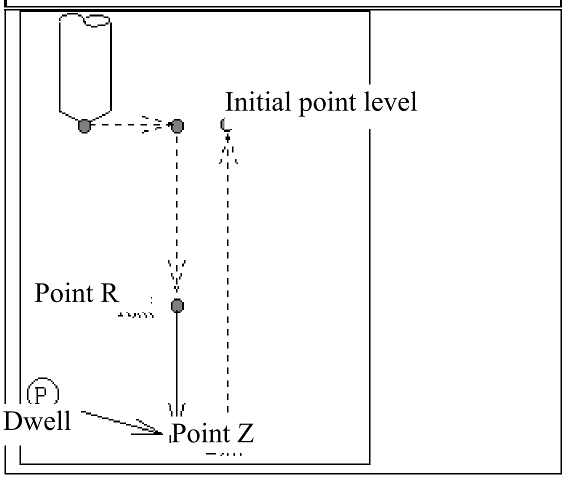
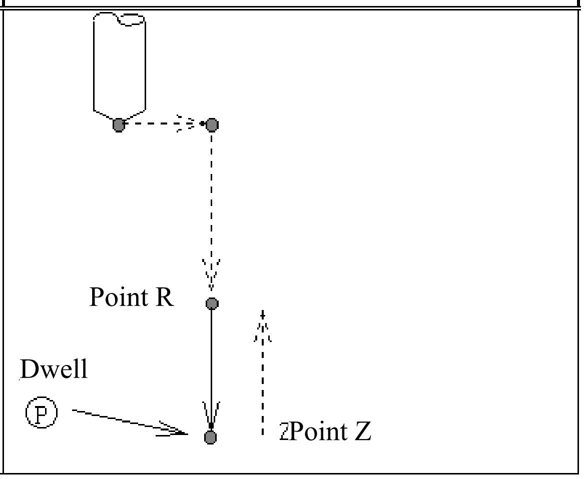
(2) Down to the point R plane at the rapid traverse;

(3) Cutting feed to the bottom of a hole

(4) Dwell for P time if it is commanded.

(5) Returning to the initial point or point R plane according to G98 or G99 at the rapid traverse;

Command Path:

G98 Return to the initial point level at the rapid traverse	G99 Return to the point R plane at the rapid traverse
	

Related Explanation:

(1) They are basically the same as G81 (drilling and spot-drilling machining), it is up after dwell at the bottom of a hole only (the dwell time is specified by P, the dwell will not be executed if it is not specified, and the command action is same as that of G81). In the blind hole, the accuracy of hole can be improved by the dwell.

(2) The command Q is disabled in this cycle, but its value will be reserved as the canned cycle modal value.

3.15.2.6 Peck drilling cycle G83

Format: G98/G99 G83 X_ Y_ R_ Z_ Q_ F_ L_ ;

Function: This cycle performs high-speed peck drilling; it performs intermittent cutting feed to the bottom of a hole while removing chips from the hole.

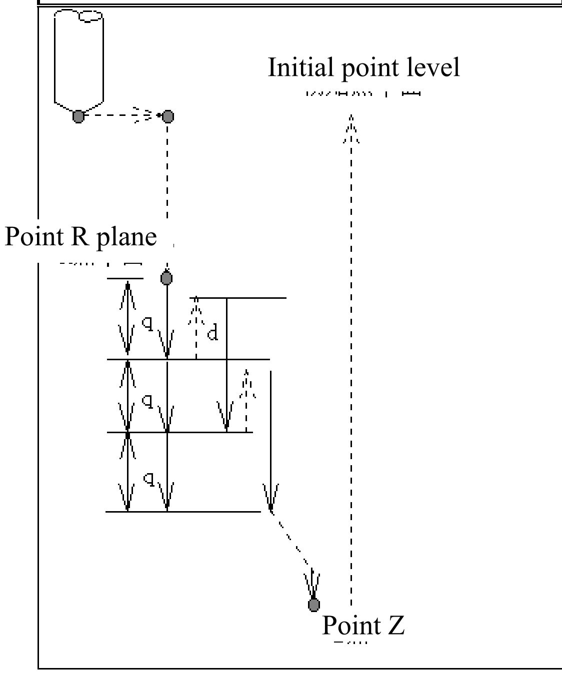
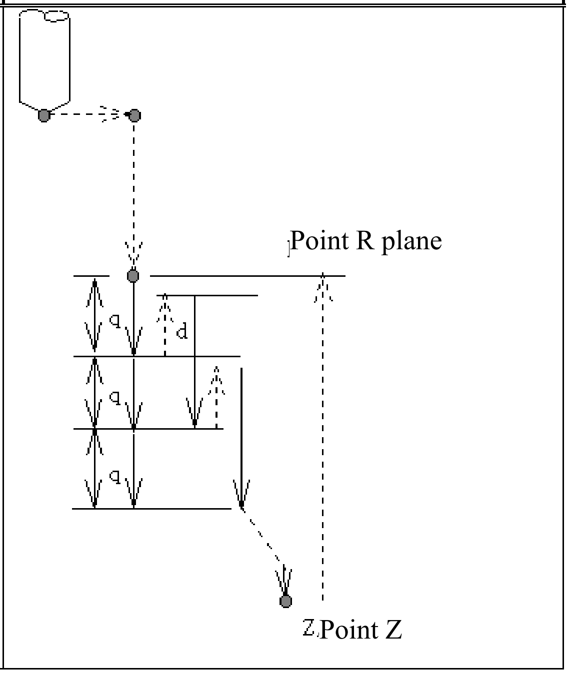
Explanation: The command explanation for canned cycle, see the table 13.1.7.

Cycle Process:

(1) Positioning to the XY plane level at the rapid traverse;

(2) Down to the point R plane at the rapid traverse;
(3) Cutting feed for Q distance;
(4) Retract to the point R plane at the rapid traverse;
(5) Rapid feed to d distance to the end surface
(6) Cutting feed for (Q+d) distance;
(7) Cycling (4) (5) and (6) to the bottom of a hole along Z-axis;
(8) Return to the initial point or point R plane according to the G98 or G99 at the rapid traverse;

Command Path:

G98 returned to the initial plane at the rapid traverse	G99 returned to the point R plane at the rapid traverse
	

Related Explanation:

(1) Same as G73, after feeding for Q, it returns to the point R plane at the rapid traverse firstly, and then rapid feeds to d mm to the end surface, then cutting feed is applied and the cycle is performed in turn. The Q value should be positive, even if the negative value is specified, and the sign is also disabled. Q value 0.001mm is defaulted if Q value is not specified; d, is set by the parameter No.52, its default value is 1000, and the unit is 0.001mm. If the cutting depth is less than the Q value, then cutting to the bottom of a hole at the first time, and rapid traverse retraction is not performed.

(2) The command P is disabled in this cycle, but its value will be reserved as canned cycle modal value.

3.15.2.7 Boring cycle G85

Format: G98/G99 G85 X_ Y_ R_ Z_ F_ L_ ;

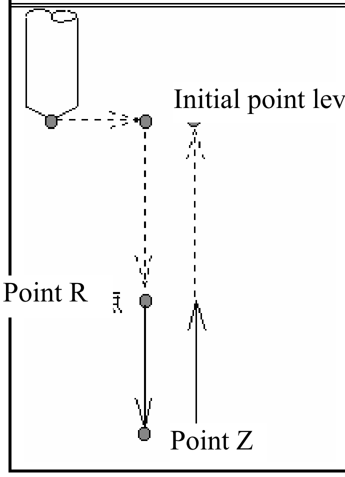
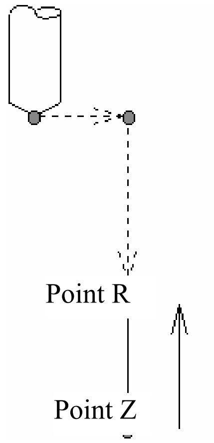
Function: After positioning along X and Y axes, rapid traverse is performed to point R; the boring is performed from point R to point Z thereafter. Cutting feed is performed to return point R plane when the Z point has been reached the bottom of a hole.

Explanation: Command explanation for the canned cycle, see the table 13.1.7.

Cycle process:

(1) Positioning to the XY plane level at the rapid traverse;
(2) Down to the point R plane at the rapid traverse;
(3) Cutting feed to the bottom of a hole;
(4) Cutting feed to the point R plane;
(5) Returning to the initial point level if it is G98;

Command Path:

G98 (Mode for returning to initial point level)	G99 (Mode for returning to point R plane)
	

Related Explanation:

(1) This cycle is used to bore a hole. The command motion is basically same as the G81 (Drilling, Spot-drilling cycle), the difference is that by the G81 it returns to the point R plane in rapid traverse rate, while by the G85 it returns to the point R plane in feedrate when the cutting feed reaches the bottom of a hole.
(2) The Q and P commands are disabled in this cycle, but its value is reserved as the canned cycle modal value.

3.15.2.8 Boring cycle G86

Format: **G98/G99 G86 X_ Y_ R_ Z_ F_ L_ ;**

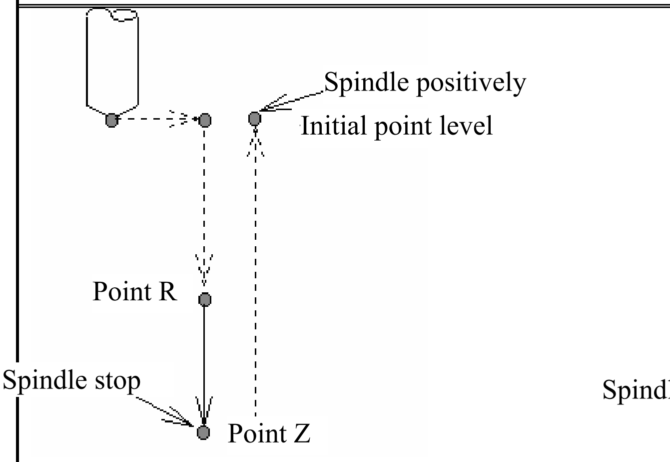
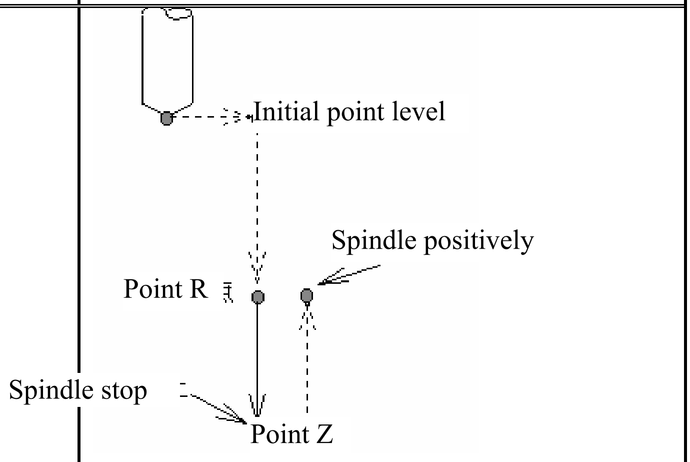
Function: After positioning along X and Y axes, rapid traverse is performed to R point, and the boring is performed from point R to point Z. The tool is retracted in rapid traverse and spindle is rotated positively when the spindle is stopped at the bottom of the hole.

Explanation: For command explanation for canned cycle, see the table 13.1.7.

Cycle process:

(1) Positioning to the XY plane level at the rapid traverse;
(2) Down to the point R plane at the rapid traverse;
(3) Cutting feed to the bottom of a hole;
(4) The spindle stops;
(5) Returning to the initial point or point R plane at rapid traverse according to the G98 or G99;
(6) The spindle is rotated in the positive direction;

Command Path:

G98 (Mode for returning to start point level)	G99 (Mode for returning to point R plane)
	

Related Explanation:

(1) This cycle is used to be bore a hole. The command operation is basically same with G81, only spindle rotation status is different. After cut feeds to the bottom of a hole, the M05 is executed (spindle stops), then the point R plane is retracted at the rapid traverse, the M03 is then performed (spindle rotates positively) regardless of the currently spindle rotation status and the positive or negative rotation are specified before the canned cycle.

(2) The command Q and P are disabled in this cycle, but its value is reserved as canned cycle modal value.

3.15.2.9 Boring cycle G88

Format: G98/G99 G88 X_ Y_ R_ Z_ P_ F_ L_ ;

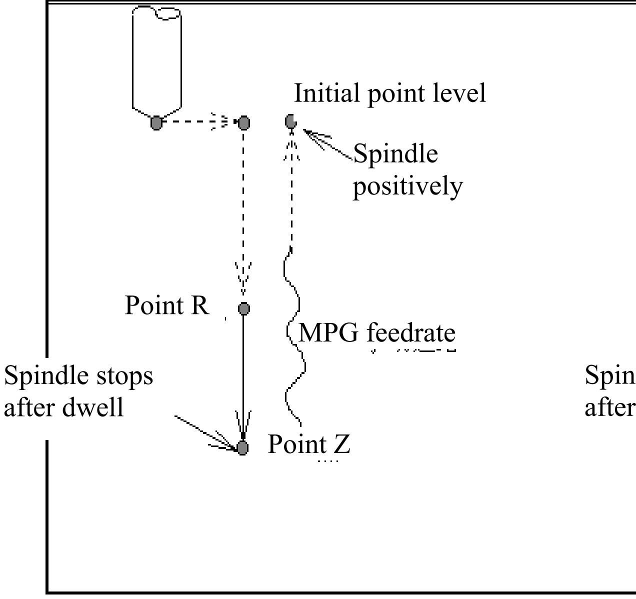
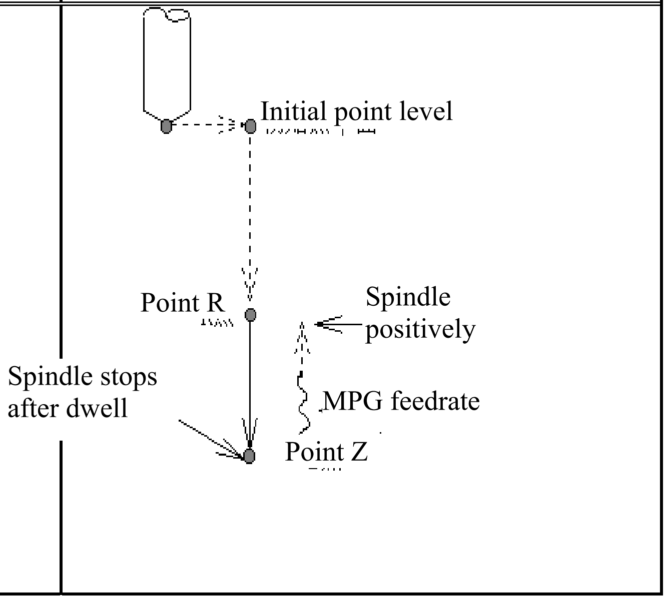
Function: A dwell is performed at the bottom of a hole, the spindle is stopping. If the manual operation is applied now, tool can be removed manually. It is better to retract the tool safely from the hole regardless of any kind of manual operation. It is rapidly retracted to point R or initial plane when the automatic operation is performed again, the spindle is stopped and G88 is finished.

Explanation: For the command explanation of the canned cycle, see the table 13.1.7.

Cycle process:

(1) Positioning to the XY plane at the rapid traverse rate;
(2) Down to the point R plane at the rapid traverse rate;
(3) Cutting feed to the bottom of hole;
(4) The spindle is stopped;
(5) P time is delayed if it is specified.
(6) Manual operation will be performed if the dwell is executed.
(7) Restoring the automatic mode, retracting to initial point or point R plane according to the G98 or G99 at the rapid traverse rate.
(8) The spindle rotates positively;

Command Path:

G98 (Mode for returning to initial plane)	G99 (Mode for returning to point R plane)
	

Related Explanation:

The command Q is disabled in this cycle, but its value is reserved as the canned cycle modal value.

3.15.2.10 Boring cycle G89

Format: G98/G99 G89 X_ Y_ R_ Z_ P_ F_ L_ ;

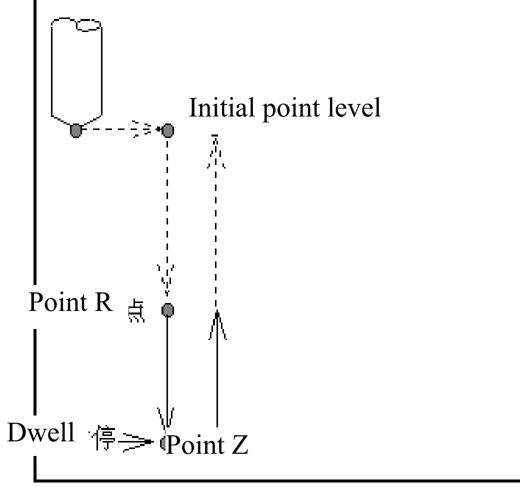
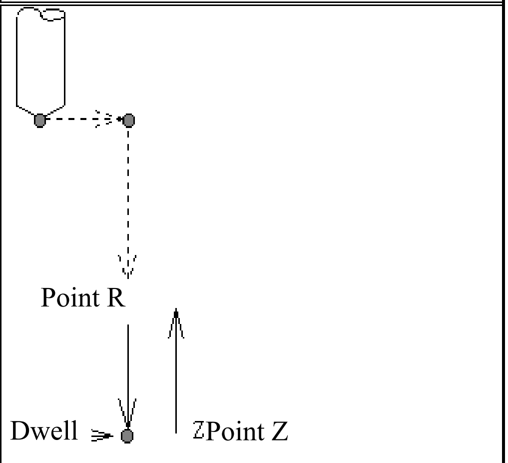
Function: This cycle is used to bore a hole normally. This cycle performs a dwell at the bottom of the hole; the tool is then retracted from the bottom of the hole at the rapid traverse rate.

Explanation: For the command explanation of the canned cycle, see the table 13.1.7.

Cycle process:

(1) Positioning to XY plane at the rapid traverse rate;
(2) Down to the point R plane at the rapid traverse rate;
(3) Cutting feed to the bottom of a hole;
(4) For dwell time P if the P is specified;
(5) Cutting feed to the point R plane;
(6) Returning to the initial point level if it is G98;
(7) Returning to the initial point or point R plane at the rapid traverse according to the G98 or G99;

Command Path:

G98 (Mode for returning to initial point level)	G99 (Mode for returning to point R plane)
	

Related Explanation:

(1) G89 (Boring cycle) is basically same as the G85, a dwell is applied at the bottom of a hole (Dwell time is specified by P, if it is not specified, the dwell is not applied, the command operation is same to the G85)

(2) The command Q is disabled in this cycle, but its value is reserved as canned cycle modal value.

3.15.2.11 Groove rough milling inside the round G110/G111

Format:

G110
G98/G99 **X_ Y_ R_ Z_ I_ W_ Q_ K_ V_ D_ F_**
G111

Function: From the beginning of the center point, arc interpolations are performed helically till the round groove of programming dimension has been machined.

Explanation: For command explanation of the canned cycle, see the table 13.1.7.

G110: Groove rough-milling inside the round in CCW;

G111: Groove rough-milling inside the round in CW;

I: I is radius inside the round groove, it should be more than the radius of current tool.

W: The firstly cutting depth is from the R reference level to the undersurface along the Z axis direction, it should be more than 0 (The first cutting position is over the bottom of the groove, then bottom position is regarded as machining position);

Q: The cutting incremental value each time along Z axis direction;

K: The width increment of cut inside XY plane, it should be less than the tool radius, and more than 0;

V: The distance to the end machining plane at the rapid traverse, it should be more than 0 when cutting;

D: Tool radius serial number, the value range is 0~32, 0 is the default of D0. The current

tool radius is determined by the specified serial number.

Cycle process:

(1) Positioning to the XY plane level at the rapid traverse rate;

(2) Down to the point R plane at the rapid traverse rate;

(3) Cut W depth downwards in cutting feedrate

(4) Mill a round face with radius I helically by K increment each time from center point to outside.

(5) The Z axis is retracted to the R reference surface at the rapid traverse rate;

(6) X and Y axes are positioned to the center at the rapid traverse rate;

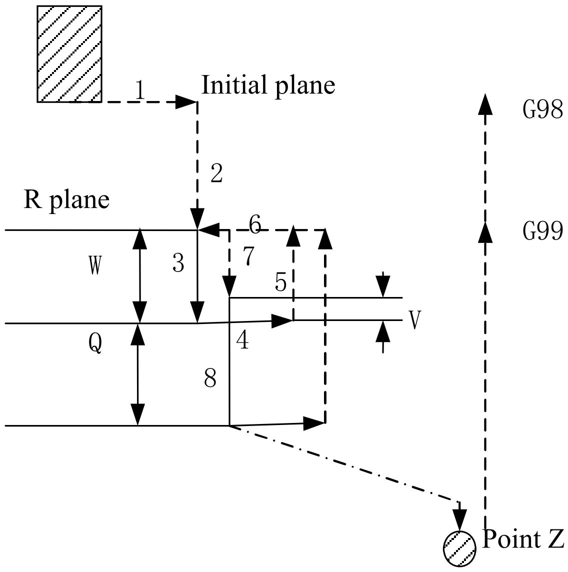
(7) Down to distance V to the end machining surface along Z axis at the rapid traverse rate;

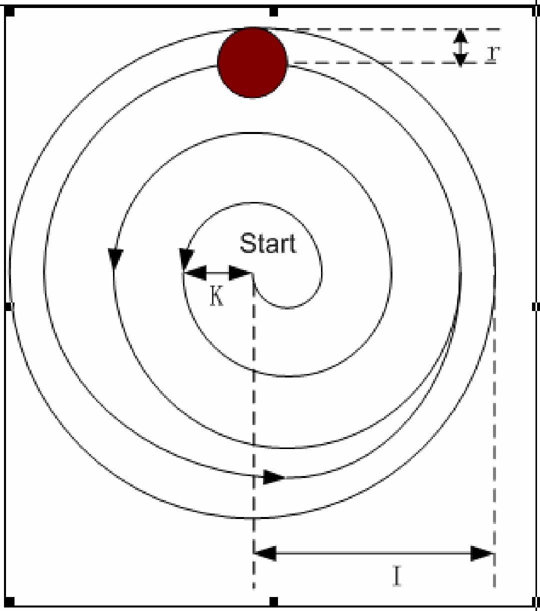
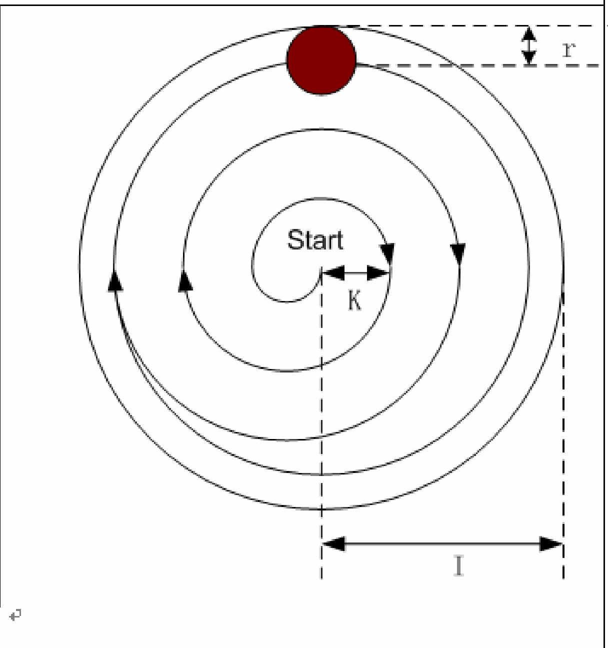
(8) Cut along Z axis for (Q+V) depth;

(9) Cycling the operations from (4) ~ (8) till the round surface of total depth is finished.

(10) Return to the initial plane or point R plane according to G98 or G99.

Command Path:

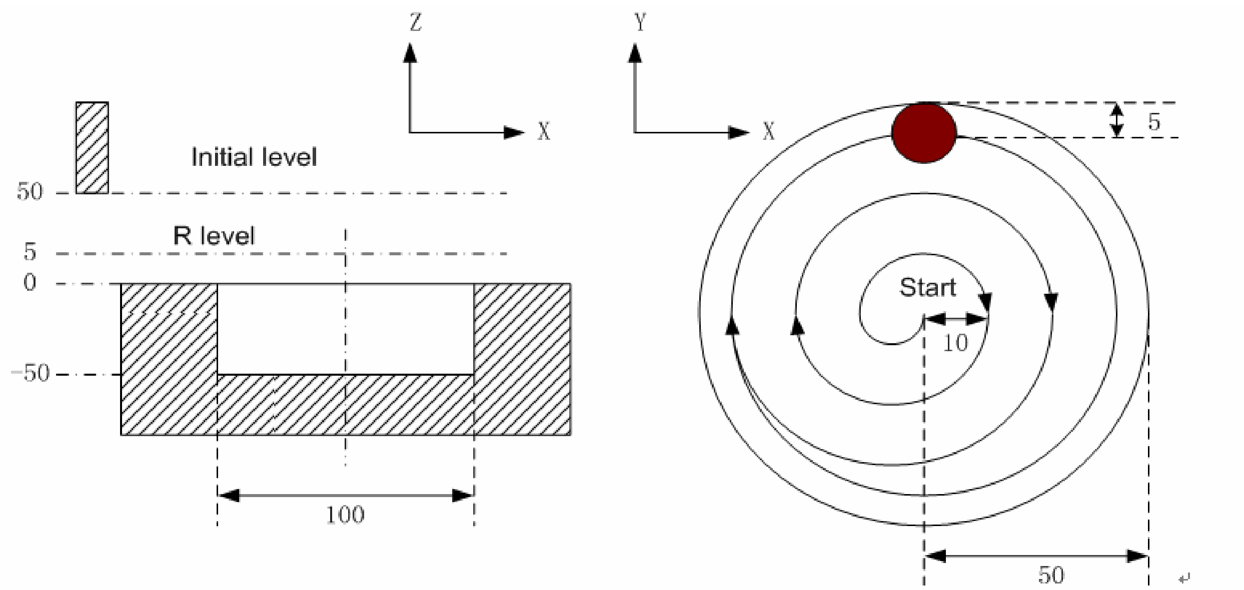


G110	G111
	

Related Explanation:

The P and L are disabled in this cycle, but the P value will be reserved as canned cycle modal value.

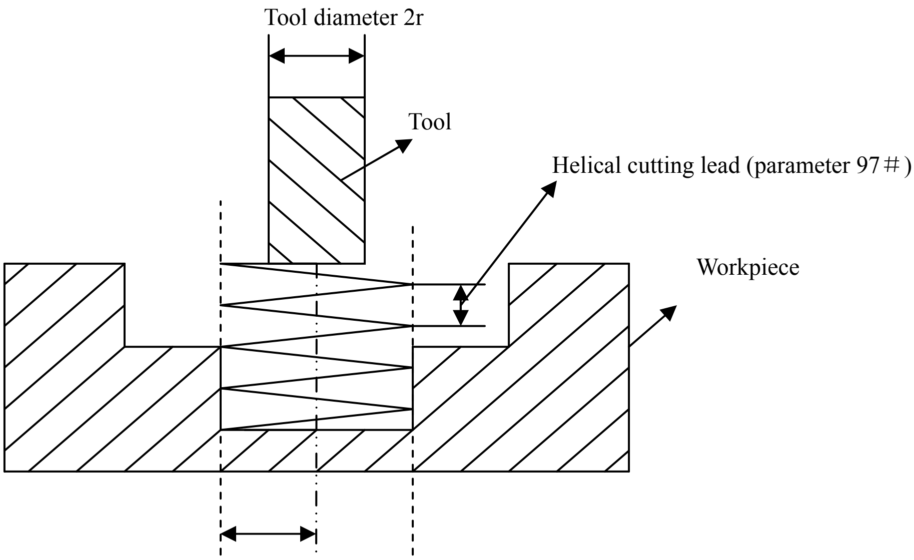
For example: A round inside groove rough-milling is specified in canned cycle G111, see the following
Figure



G90 G00 X50 Y50 Z50; (G00 positioning at the rapid traverse rate)
G99 G111 X25 Y25 R5 Z-50 150 W20 Q10 K10V10 F800 D1; (Rough-milling cycle inside the round groove D1=5)
G80 X50 Y50 Z50; (Canceling canned cycle, returning from the point R plane)
M30;

Note: Set the 97# parameter value to one which is more than 10, by G110 and G111 it feeds helically along Z axis. Rough-milling machining can be directly performed for non-groove workpiece.

See the following figure for helical cutting path:



3.15.2.12 Fine-milling cycle inside full circle G112/G113

Format:

	G112	
G98/G99		**X_ Y_ R_ Z_ I_ J_ D_ F_**
	G113	

Function: A fine-milling inside the full circle is finished with the specified radius value I and direction, the tool is retracted after the fine-milling.

Explanation: For command explanation of canned cycle, see the table 13.1.7.

G112: Fine-milling cycle inside the full circle in CCW.

G113: Fine-milling cycle inside the full circle in CW

I: Fine-milling circle radius, the value range is indicated as 0~9999.999mm, the absolute value is taken when it is negative.

J: Fine-milling distance from start point to the center point, the value range is indicated as 0~9999.999mm, the absolute value is taken when it is negative

D: Sequence number of tool radius, the value range is indicated as 0~32, the 0 is default of D0. The current tool radius value is taken according to the specified sequence number.

Cycle process:

(1) Positioning to the XY plane level at the rapid traverse rate;

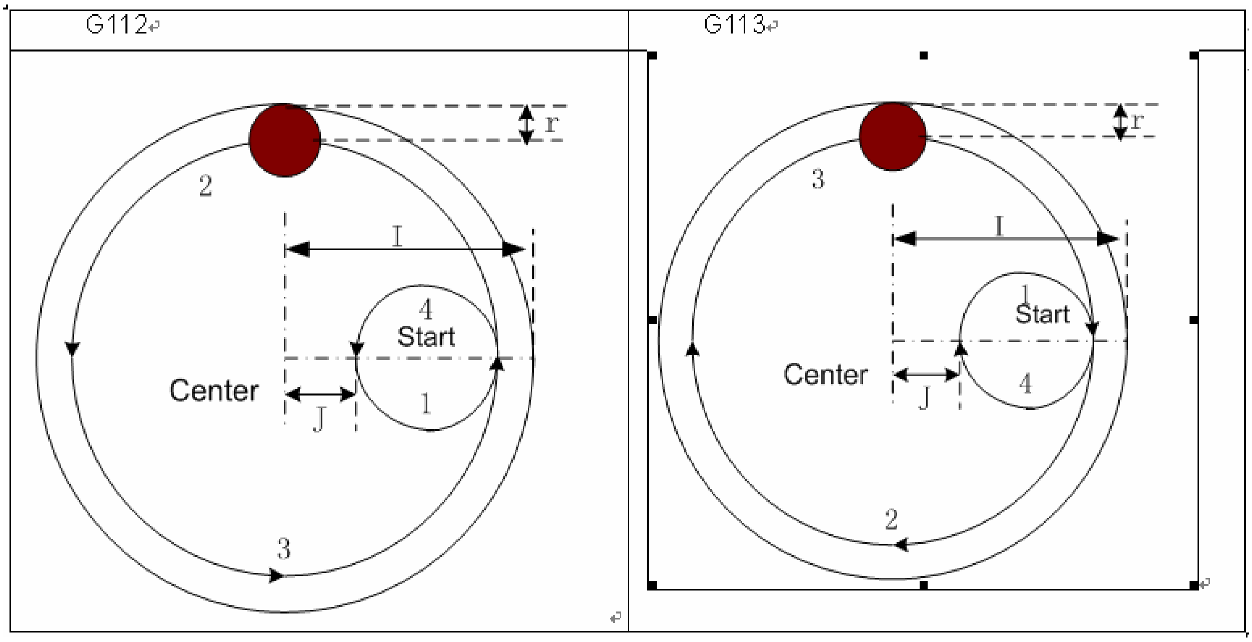
(2) Down to the point P level at the rapid traverse rate;

(3) Feed to the bottom of a hole;

(4) Perform the circle interpolation by the path of transit arc 1;

(5) Perform the full circle interpolation by the path of arc 2 and arc 3;
(6) Perform circular interpolation by the path of transit arc 4 and return to the start point;
(7) Return to the initial point level or point R plane according to G98 or G99.

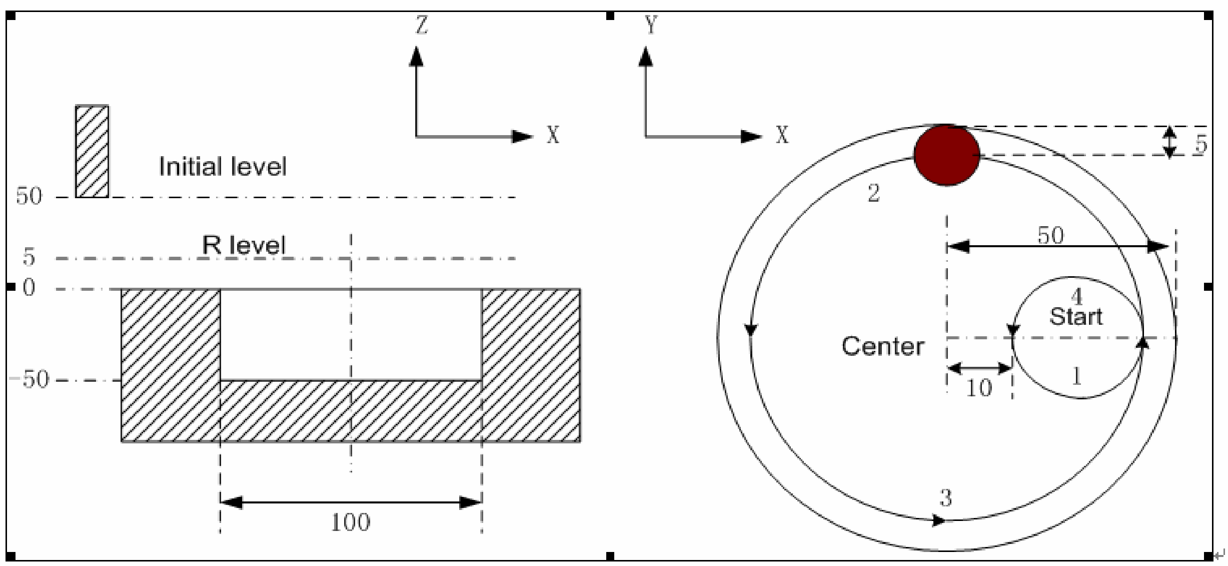
Command Path:



Related Explanation:

The commands Q, P and L are disabled in this cycle, but the Q and P value will be reserved as the canned cycle modal value.

For example: Fine-mill a finished rough-milling round groove by the canned cycle G112 command, see the following figure:



G90 G00 X50 Y50 Z50; (G00 rapid positioning)
G99 G112 X25 Y25 R5 Z-50 150 J10 F800 D1; (Start canned cycle, fine-milling cycle

inside the circle at the bottom of a hole D1=5)

G80 X50 Y50 Z50; (The canned cycle is cancelled, returning from the point P level)

M30;

3.15.2.13 Fine-milling cycle outside circle G114/G115

Format:

G114

G98/G99 X_ Y_ R_ Z_ I_ J_ D_ F_;

G115

Function: A fine-milling outside the full circle is performed by the specified radius value and the direction, and the tool is retracted after the fine-milling is finished.

Explanation: For command explanation of canned cycle, see the table 13.1.7.

G114: Finish-milling cycle for outside circle in CCW.

G115: Finish-milling cycle for outside circle in CW.

I: A fine-milling circle radius, the value range is indicated as 0~9999.999mm, the absolute value is taken when it is negative.

J: Distance of fine-milling between the start point and the circle, the value range is indicated as 0~9999.999mm; the absolute value is taken when it is negative.

D: The sequence number of tool radius, the value range is 0~32, 0 is the default of D0. The current tool radius value is taken according to the specified sequence number.

Cycle process:

(1) Positioning to the XY plane level at the rapid traverse rate;

(2) Down to the point R plane at the rapid traverse rate;

(3) Cutting feed to the bottom of a hole;

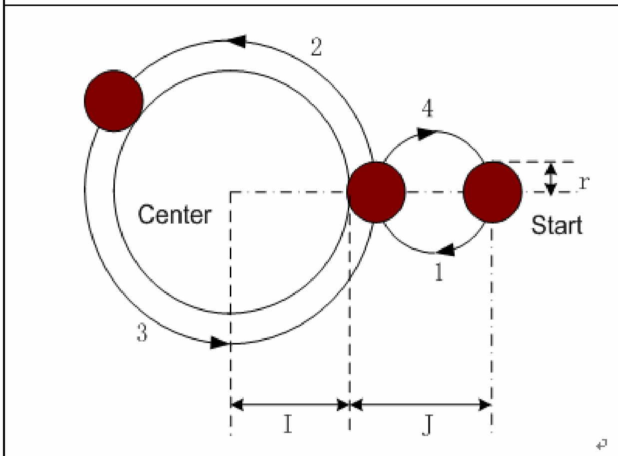
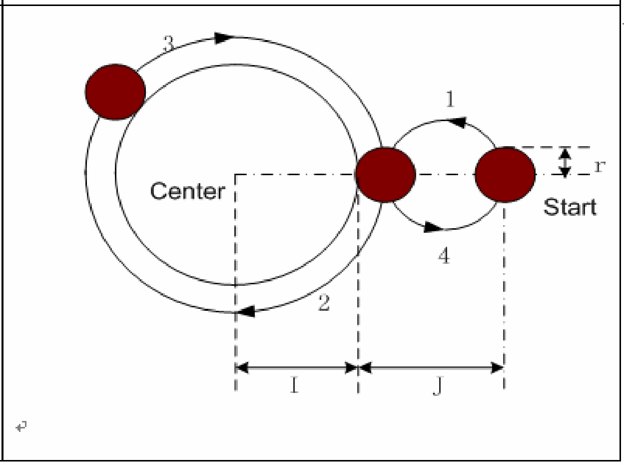
(4) Perform the circle interpolation by the path of transit arc 1;

(5) Perform the full circle interpolation by the path of arc 2 and arc 3;

(6) Perform circular interpolation by the path of transit arc 4 and return to the start point;

(7) Return to the initial point level or point R plane according to G98 or G99.

Command path:

G114	G115
	

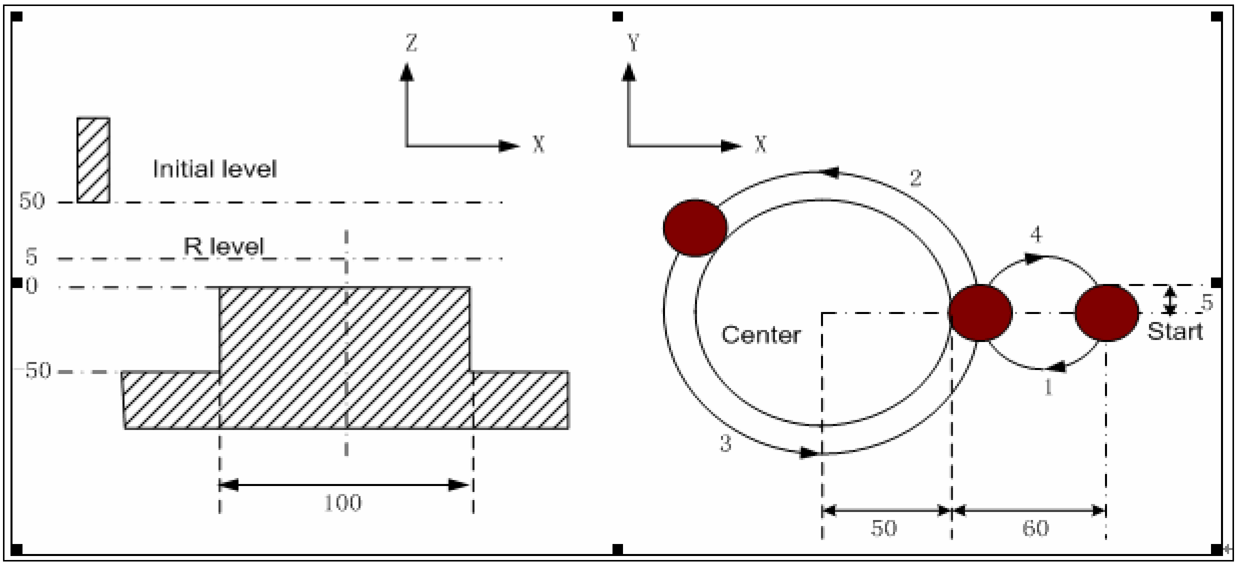
Related Explanation:

(1) The interpolation direction of between transit arc and fine-milling arc are different when the fine-milling outside circle is performed, the interpolation direction in command explanation is

the interpolation direction of fine-milling arc.

(2) The command Q, P and L are disabled in this cycle, but the Q and P value are reserved as canned cycle modal value.

For example: A finished rough-milling round groove is performed by fine-milling with the canned cycle G114 command, see the following figure :



G90 G00 X50 Y50 Z50;　　(G00 rapid positioning)

G99 G114 X25 Y25 R5 Z-50 150 J60 F800 D1;　　(Start canned cycle, the fine-milling cycle is performed outside the circle at the bottom of a hole D1=5)

G80 X50 Y50 Z50;　　(The canned cycle is cancelled, returning from the point R plane)

M30；

3.15.2.14 Rectangle groove rough-milling　G134/G135

Format: **G134　G98/G99**
X_　Y_　Z_　R_　I_　J_　K_　W_　Q_　V_　U_　D_　F_
G135

Function: From the center of the rectangle, the linear cutting cycle is applied by the specified parameter
data, till the rectangle groove with programmed dimension is made out.

Explanation: For command explanation of canned cycle, see the table 13.1.7.

G134: Rectangle groove rough-milling in CCW

G135: Rectangle groove rough-milling in CW

I: The width of rectangle groove along the X axis direction

J: The width of rectangle groove along the Y axis direction.

K: The cut width increment inside XY plane, it is less than the tool radius, but, more than 0.

W: For the first cutting along the Z axis direction, the distance is downward to the R reference surface, it is more than 0 (if the first cutting is over the position of the bottom of the groove, then the bottom of the groove is taken as the machining position)

Q: The cutting incremental value each time along Z axis.

V: Distance to the end machining surface, which is more than 0, when the rapid traverse

is executed.

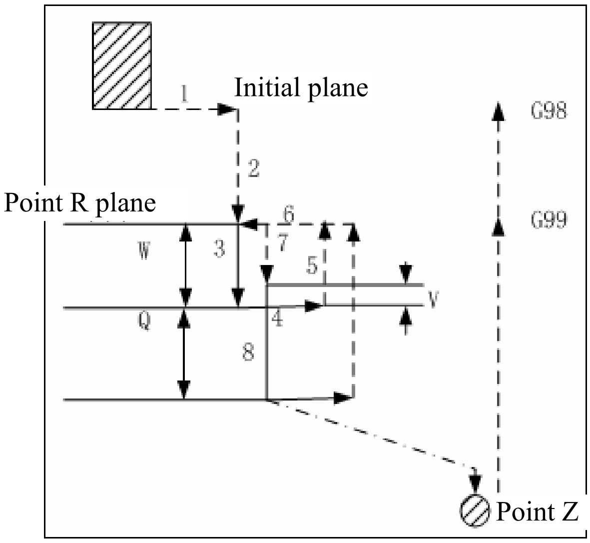
U: Corner arc radius, if it is omitted, that is no corner arc transition is not shown.

D: Sequence number of tool radius, its value range is indicated as 0 ~ 32, thereunto, the 0 is default of D0. The current tool radius value is taken out according to the specified sequence number.

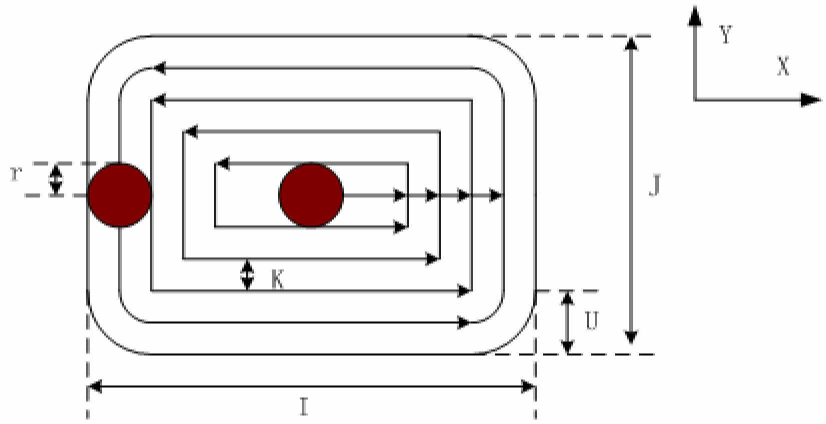
Cycle process:

(1) Positioning to the XY plane at the rapid traverse rate;
(2) Down to the point R plane at the rapid traverse rate;
(3) W distance depth is cut downwards by cutting feedrate
(4) Mill a rectangle face helically by K increment each time from center point to outside.
(5) R reference surface is retracted along the Z axis at the rapid traverse rate.
(6) The center of rectangle is positioned along the X and Y axes at the rapid traverse rate.
(7) Down to distance V to the end machining surface along Z axis at the rapid traverse rate;
(8) Cut along Z axis for (Q+V) depth;
(9) Cycling the operation from (4) ~ (8) till the surface of total cutting is performed.
(10) Return to the initial plane or point R plane according to G98 or G99.

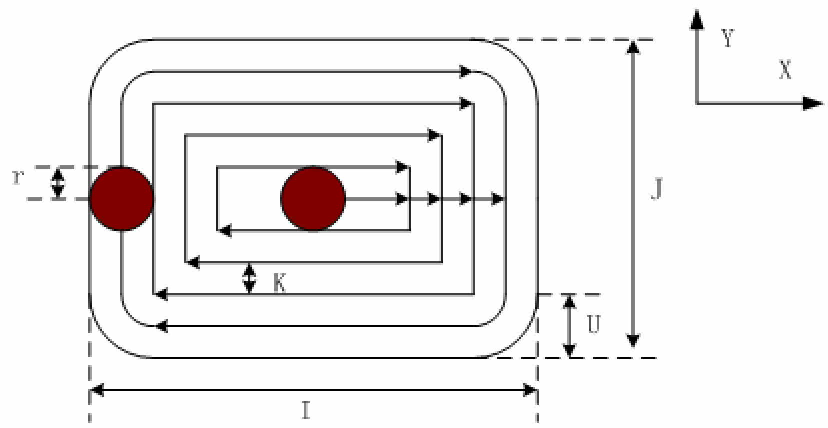
Command Path:



G134



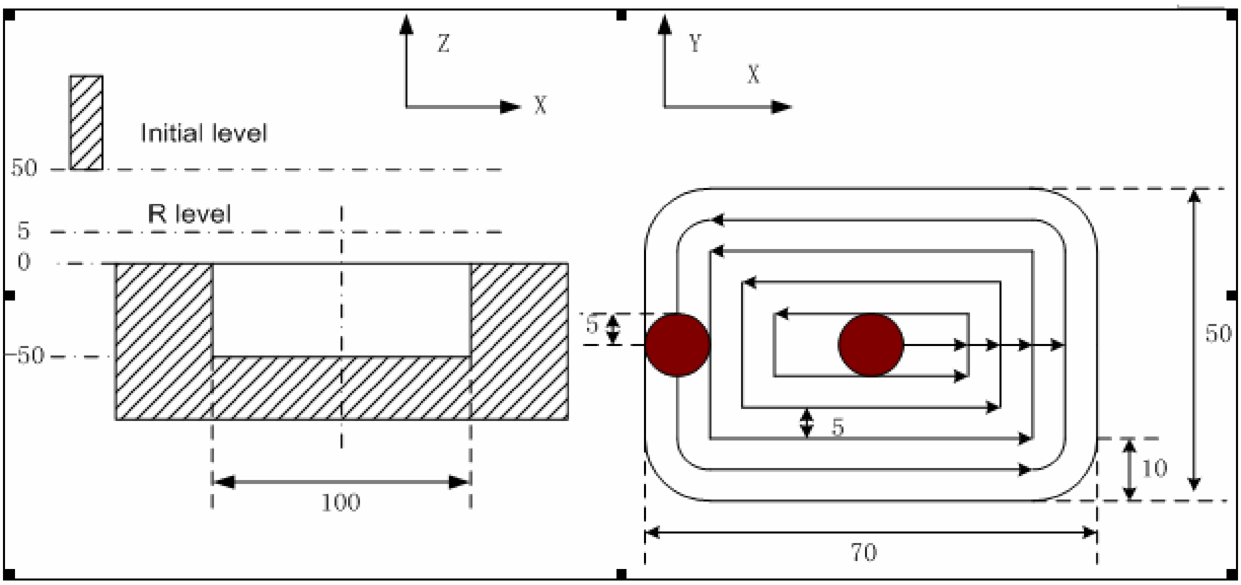
G135



Related Explanation:

The commands P and L are disabled in this cycle, but the P value is reserved as canned cycle modal value.

For example: An inside rectangle groove rough-milling is specified by G134 in canned cycle, see the following figure:



G90 G00 X50 Y50 Z50; (G00 rapid positioning)

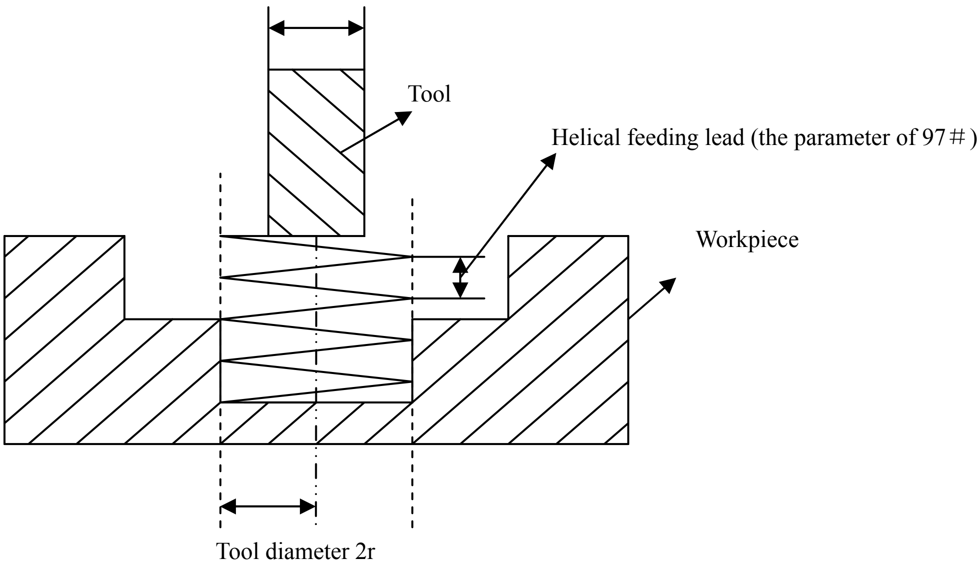
G99 G134 X25 Y25 R5 Z-50 I70 J50 W20 Q10 K5 V10 U10 F800 D1; (Groove rough-milling cycle inside rectangle is performed D1=5)

G80 X50 Y50 Z50; (The canned cycle is cancelled, returning from the point R plane)

M30;

Note If the parameter value of 97# is set for more than 10, the helical cutting feed along the Z axis will be performed by G110 and G111. So, the workpiece without groove can be machined by rough-milling directly.

The helical feeding path is as follows:



3.15.2.15 Rectangle groove inner fine-milling cycle G136/G137

Format:

G136
G98/G99 X_ Y_ R_ Z_ I_ J_ D_ K_ U_ F_;
G137

Function: The tool performs fine-milling inside the rectangle with the specified width and direction, it is returned after finishing the fine-milling.

Explanation: For command explanation of canned cycle, see the table 13.1.7.

G136: Finish-milling cycle inside groove of rectangle in CCW.

G137: Finish-milling cycle inside groove of rectangle in CW.

I: The rectangle width along the X axis, the value range is indicated as 0~9999.999mm.

J: The rectangle width along the Y axis, the value range is indicated as 0~9999.999mm.

D: Sequence number of tool radius, the value range is 0~32, the 0 is default value of D0. The current tool radius value is taken out according to the specified sequence number.

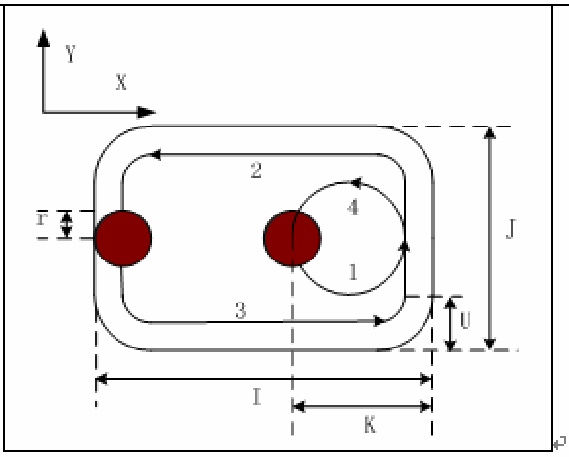
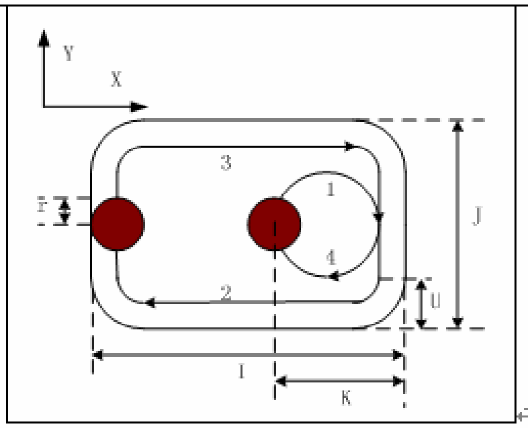
K: The distance between the finish-milling start point and the rectangle side in X axis direction, the value range is indicated as 0~9999.999mm.

U: Corner arc radius; no corner arc transition if it is omitted. When the U is omitted or it is equal to 0 and the tool radius is more than 0, the alarm is generated.

Cycle process:

(1) Positioning to XY plane at the rapid traverse rate;
(2) Down to point R plane at the rapid traverse rate;
(3) Cutting feed to the bottom of a hole;
(4) Perform the circle interpolation by the path of transit arc 1;
(5) Perform the circular and linear interpolation by the path of 2-3-4-5-6;
(6) Perform circular interpolation by the path of transit arc 7 and return to the start point;
(7) Returning to the initial plane or point R plane according to G98 or G99.

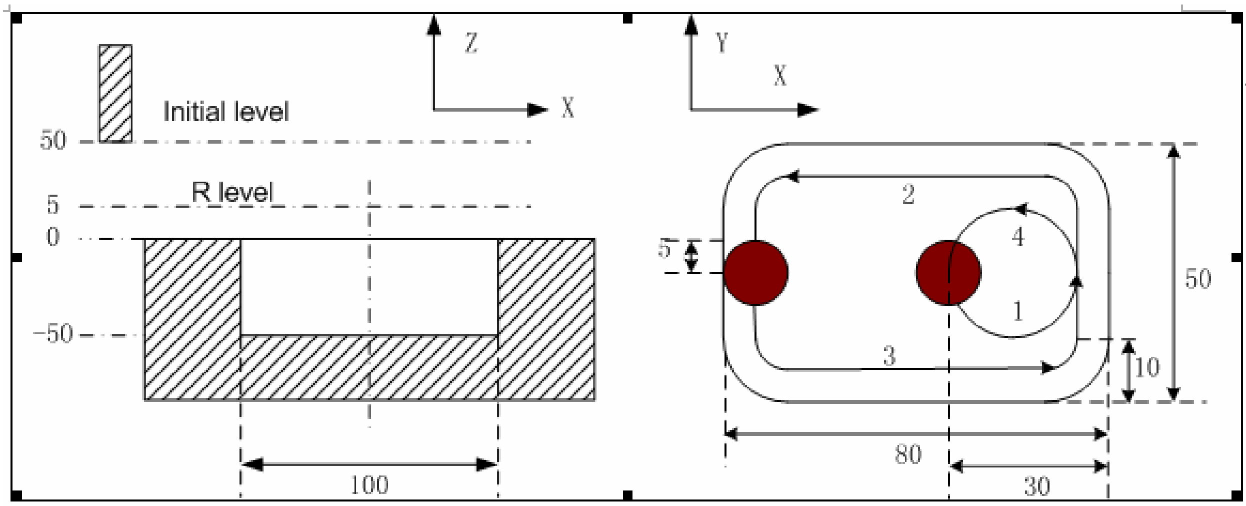
Command Path:

G136	G137
	

Related Explanation:

The commands Q, P and L are disabled in this cycle, but the Q and P values are reserved as the canned cycle modal value.

For example: To perform a fine-milling for the finished rough-milling rectangle groove with the canned cycle G136 command, see the following figure:



G90 G00 X50 Y50 Z50; (G00 rapid positioning)
G136 X25 Y25 R5 Z-50 I80 J50 K30 U10 F800 D1; (Perform finish-milling inside the rectangle groove at the bottom of a hole in the canned cycle D1=5)
G80 X50 Y50 Z50; (The canned cycle is cancelled, returning from the point R plane)
M30;

3.15.2.16 Finish-milling cycle outside the rectangle G138/G139

Format:

G98/G99 **G138** / **G139** **X_ Y_ R_ Z_ I_ J_ D_ K_ U_ F_**

Function: The tool performs fine-milling outside the rectangle by the specified width and direction, it is returned after finishing the fine-milling.

Explanation:

G138: Finish-milling cycle outside the rectangle in CCW.

G139: Finish-milling cycle outside the rectangle in CW.

I: The width of rectangle along the X axis, the value range is indicated as 0~9999.999mm.

J: The width of the rectangle along the Y axis, the value range is indicated as 0~9999.999mm.

D: Sequence number of tool radius, its value range is indicated as 0 ~ 32, thereinto, the 0 is default of D0. The current tool radius value is taken out according to the specified sequence number.

K: The distance between the finish-milling start point and the side of rectangle along the X axis, the value range is indicated as 0~9999.999mm.

U: Corner arc radius, if it is omitted, no corner arc transition.

Cycle process:

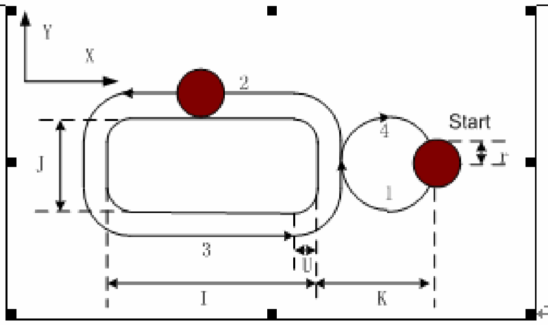
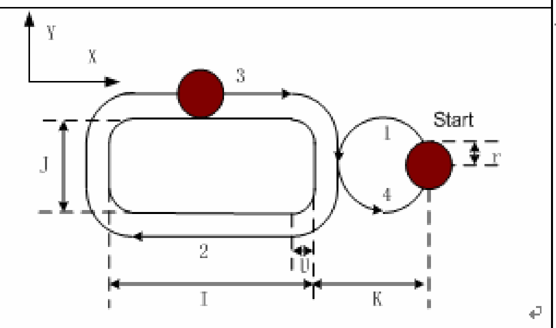
(1) Positioning to the XY plane at the rapid traverse rate;
(2) Down to the point R plane at the rapid traverse rate;
(3) Cutting feed to the bottom of a hole;
(4) Perform the circle interpolation by the path of transit arc 1;

(5) Perform the circular and linear interpolation by the path of 2-3-4-5-6;

(6) Perform circular interpolation by the path of transit arc 7 and return to the start point;

(7) Returning to the initial plane or point R plane according to G98 or G99.

Command Path:

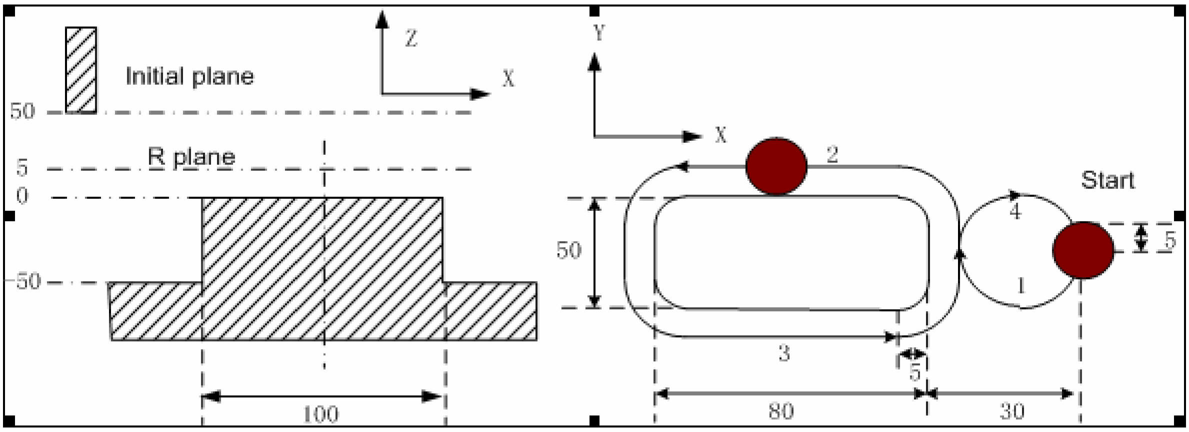
G138	G139
	

Related Explanation:

(1) The interpolation direction of transition arc is inconsistent to that of the fine-milling arc when a fine-milling is performed outside the rectangle. The interpolation direction is the one for the fine-milling arc in the command explanation.

(2) The commands Q, P and L are disabled in this cycle, but, the value of Q and P are reserved as canned cycle modal value.

For example: A finished rough-milling rectangle groove is performed by the fine-milling by the command G138 in canned cycle. See the following figure.



G90 G00 X50 Y50 Z50; (G00 rapid positioning)

G99 G138 X25 Y25 R5 Z-50 180 J50 K30 U5 F800 D1; (The rectangle outside finish milling is performed under the canned cycle at the bottom of a hole D1=5)

G80 X50 Y50 Z50; (The canned cycle is cancelled, it returns from the point R plane)

M30;

3.15.3 Continous Drilling

Continuous equal interval drilling cycle is performed in the way that canned cycle is called according to the specified linear, rectangular or arc path.

Parameters related to continuous drilling

0	1	5

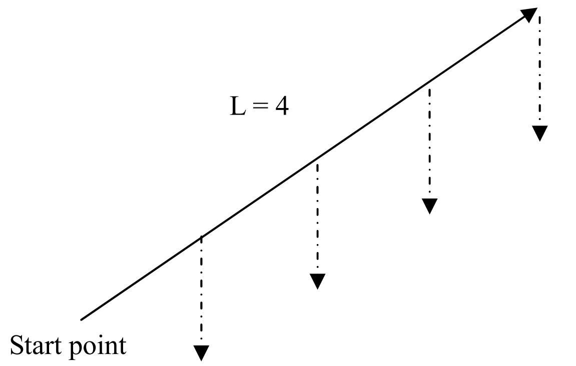
LPTK	RPTK		BRCH	***	***	***	***

LPTK =1：Locating with G01 in line interval drill；
=0：Locating with G00 in line interval drill；
RPTH =1: Locating with G01 in circle and rectangle interval drill；
=0：Locating with G00 in circle and rectangle interval drill；

BRCH =1：the return plane when continuous drilling is selected by G98, G99.
=0：the return plane when continuous drilling is selected by G99.

3.15.3.1 Line series punch (L function)

L holes machining cycle should be performed from current plane position to end point specified by X and Y are indicated if the L word is specified in canned cycle, so the current position (block start and end) will not be drilled, the end point position is regarded as the last hole, holes are equal-spaced, as follows:



L value setting	System execution result
Value is negative	Ineffective, the value should be positive
The value is unspecified or equals to 1	Normal drilling cycle 1 time
The value is 0	No change of axes, the system reserves relevant cycle modal data
The value is decimal	When L＞1,using round number When L＜1, it is processed as L=0, not moving but reserving its modal data and relevant cycle parameter values.

Note 1：the maximum input value of command L is -9999.999～9999.999; Decimals is ignored and absolute value is used instead of negative value. L code is effective only in current block.

Note 2：In continuous drilling, the return planes are R point plan. After the last hole is processed, the return plane is specified by G98/G99.

Note 3：When there is no axis position command in the specified L block, it means drilling cycle is performed L times in the original place.

Note 4：Canned cycle command G110, G111, G112, G113, G114, G115, G134, G135, G136, G137, G138, G139 has no continuous drilling function.

Note 5：When L0 is specified, no drilling will be performed.

3.15.3.2 Rectangle series punch (G140/G141)

Format:

G98/G99 G140/G141 Gxx X_ Y_ R_ Z_ A_ B_ J_ F_

Function: Performing series punch on each side of the rectangle according to the punch number specified.

Explanation:

G140 – Punching in CW

G141 – Punching in CCW

Gxx – Punching type (G73, G74, G81, G83, G84, G85, G86, G88, G89)

X, Y – End coordinate of the first rectangle side

R – R plane position

Z – Hole depth

A – The punching number on the 1st and 3rd side

B – The punching number on the 2nd and 4th side

J- The length of the 2nd side

F – Cutting feedrate

Related Parameter:

Bit 7 of the parameter 014

1: Hole positioning of serial punching is performed by cutting path (G01~G03).

0: Hole positioning of serial punching is performed by the rapid traverse path (G00).

For example:

The end point coordinate of the rectangle first side is X90, Y40; the length of the 2nd side is 20mm as for the rectangle path punching. The punching holes are machined by G81, to punch 3 holes at 1st and 3rd side each other; punch 2 holes at 2nd and 4th side each other, the hole depth is 25mm;

Its programming is as follows:

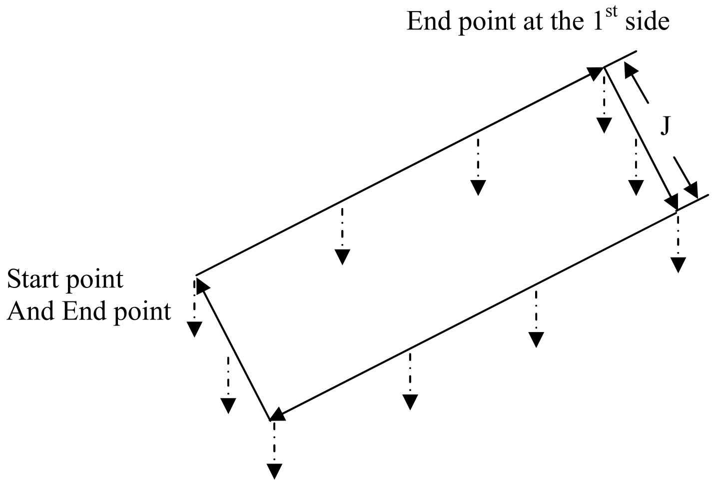
G90 G17 G0 X0 Y0 Z25；

M03；

G140 G81 X90 Y40 R5 Z-25 A3 B2 J20 F800；

G80 G0 X100 Y100 M05；

M30



There are 10 holes such as A1~A3, B4, B5, A6~A8, B9 and B10 to be machined as in above figure.

Note 1: If the G140 or G141 is specified in the canned cycle, it is indicated that the rectangle serial punching will be performed. The rectangle data are defined according to specified X, Y coordinates and J value in a program, and the serial punching cycle is performed

according to the punch mode (canned cycle command).

Note 2: The command value of maximum punching number A and B at each side is 9999; the command is disabled when it is negative. The decimal part will be rounded off if the command is decimal; if the A or B is not specified, then 0 is a default.

Note 3: The rectangle is defined by the current start point, the end of the 1st side and the length of the 2nd side; the default is current start point if the end of 1st side is not specified; the alarm will be generated if the length (namely, the J is not specified) of 2nd side is not specified.

Note 4: The returned levels are all R point plane in serial punching, the corresponding plane will be retracted according to G98/G99 specified in a block when the last hole is performed.

Note 5: Canned cycles, such as G110, G111, G112, G113, G114, G115, G134, G136, G137, G138 and G139 have no serial punching functions.

Note 6: The command words G140, G141, A, B and J are only effective in current block. The alarm will be generated if the G140 and G141 are specified without the canned cycle (punching). The A, B and K will be ignored if A, B and K are specified instead of the G140 or G141.

3.15.3.3 Arc serial punching (G142/G143)

Format:

G98/G99 {G142 / G143} Gxx X_ Y_ R_ Z_ B_ (I_ J_) C_ F_

Function: Serial punching is performed according to the specified punching number on specified arc.

Explanation:

G142 – Punching in CW

G143 – Punching in CCW

Gxx – Punching type（G73, G74,G81,G82, G83, G84,G85, G86, G88,G89）

X,Y – End point coordinate for the arc, it is fixed for G17 plane.

R – R plane position

Z – Hole depth

B – Radius of arc, when a negative value is specified, it is major arc.

(I_ J_) – The circle center and radius are calculated by I or J when the R value is not specified.

C – Number of punching

F – Cutting feedrate

Related Parameter:

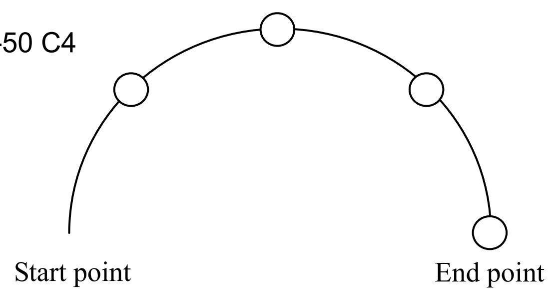
Bit 7 of the parameter 014

1: Hole positioning for serial punching is performed by cutting path (G01~G03).

0: Hole positioning for serial punching is performed by the rapid traverse path (G00).

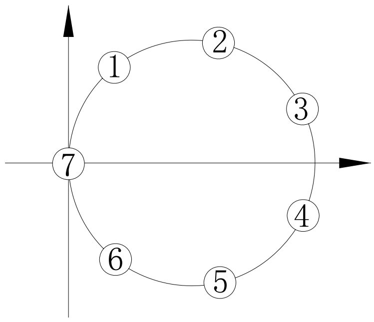
For example:

G91 G142 G81 X100 R50 Z-50 C4



Example 2：when drilling 7 holes in full circle, the start points and end points are coordinate origins, and the radius is 50, hole depth is 50.

O0001；
G00 G90 X0 Y0 Z0 G17；
G98 G142 G82 I50 J0 R-10 Z-50 C7 F3000；
M30；
%



Note 1: In continuous drilling, when the start point is identical to end point, no drilling will be performed.

Note 2: Canned cycle G110, G111, G112, G113, G114, G115, G134, G135, G136, G137, G138, G139 has no continuous drilling function.

Note 3: The maximum drilling number C is 9999; the negative value is processed as absolute value; the decimals are rounded.

Note 4：When C is not specified or equals to 0, it reaches the end point directly and no drilling will be performed.

3.15.4 Cautions for canned cycle

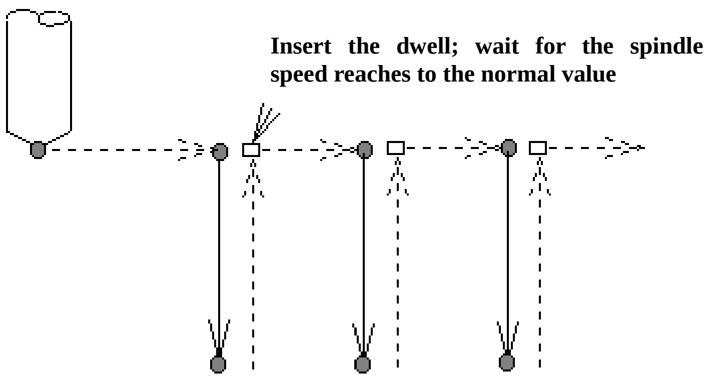
(1) The spindle should be rotated (The M code should be correctly specified, or, the alarm will be generated, the G74 by M04, G84 by M03) by using the miscellaneous function (M code) before the canned cycle is executed.

(2) Specifying any command of the X, Y, Z and R data, the hole machining can be performed in the canned cycle of G73~G89. If neither data is contained in the block, the hole machining is not performed (G110, G111, G112, G113, G114, G115, G134, G135, G136, G137, G138 and G139 are still needed to specify the corresponding address I, J and K, or the alarm occurs). But the hole machining is not performed when the G04 X_ is specified in the circumstance of X, because the X indicates for time when the G04 is specified.

G00 X_;	(G00 rapid positioning)
G81 X_ Y_ Z_ R_ F_ L_;	(Hole machining performs)
;	(Without hole machining)
F_ ;	(F value is refreshed without the hole machining)
M_ ;	(Performing the miscellaneous function only)

(3) When the canned cycle (G74 or G84) is employed in spindle rotation consolation, if the hole

position (X, Y) or distance from initial point level to the point R plane is short, and it is necessary to machine serially, or sometimes the spindle can not reach the specified speed before the hole machining operation, for delaying the time, the dwell block by G04 is inserted into each hole machining, which is shown as follows:



G86 X_ Y_ Z_ R_ F_ ;
G04 P _; (For dwell time P, without hole machining)
X_ Y_; (The next hole is machined)
G04 P _; (For dwell time P, without hole machining)
X_ Y_; (The next hole is machined)
G04 P_; (For dwell time P, without hole machining)

Sometimes, this issue will not be considered according to different machine tool, refer to the manual supplied by the machine tool builder.

(4) As stated above, the canned cycle can also be cancelled only when G00~G03 codes are read. So, there are two cases (# expresses for 0~3, □□ for canned cycle code) will be shown when they share the same block with the canned cycle G code.

G# G□□ X- Y- Z- R- Q- P- F- K-; (For canned cycle)

G□□ G# X- Y- Z- R- Q- P- F- K-; The X, Y and Z axes are moved by G#, the R, P, Q and K are disabled, the F is stored. The principle, which the last G code is effective when G codes of same group share the same block, is met by cases above.

(5) When the canned cycle and miscellaneous function are specified at the same block, The M and MF codes are delivered at the beginning of positioning (see the Fig.13.1 (A) for the operation 1). The next hole machining can be performed till the ending signal (FIN) occurs.

(6) When the canned cycle is applied, if the tool compensation C is current state, the tool compensation information C is then temporarily cancelled and saved; the tool compensation C status is restored when the canned cycle is cancelled.

(7) If the tool length offset commands (G43, G44 and G49) are specified in a canned cycle block. Then, the offset is performed when the point R plane is positioned (operation 2). The tool length offset commands are disabled after the canned cycle is entered till it is cancelled.

(8) The cautions for the operation of canned cycle:

a. Single block

When the canned cycle operation is performed by using the single block mode, normally, it is separately stopped at the terminal of the movements 1, 2, 3, 4, 5 and 6 in the Fig. 13.1 (A). And the single block is somewhat different according to corresponding canned cycle action at the bottom of a hole. For example, the single block is stopped when the dwell is applied. The operation at the bottom of the hole for fine-milling and rough-milling are divided into multiple single stop. So, it is necessary to startup for several times to machine a hole in a single block.

b. Feed hold

The feed hold is disabled between the movement 3 ~ 5 in commands G74 and G84, but the indicator of feed hold will light up. But the control stops till the operation 6. If the feed hold is performed again in operation 6, then it is stopped immediately.

c. Override

The feedrate override is considered for 100 percent in the operation G74 and G84, the override change is disabled.

(9) When the bit 1 of parameter 3 (D_R) is set to 1, the D value in tool compensation page indicates diameter value.

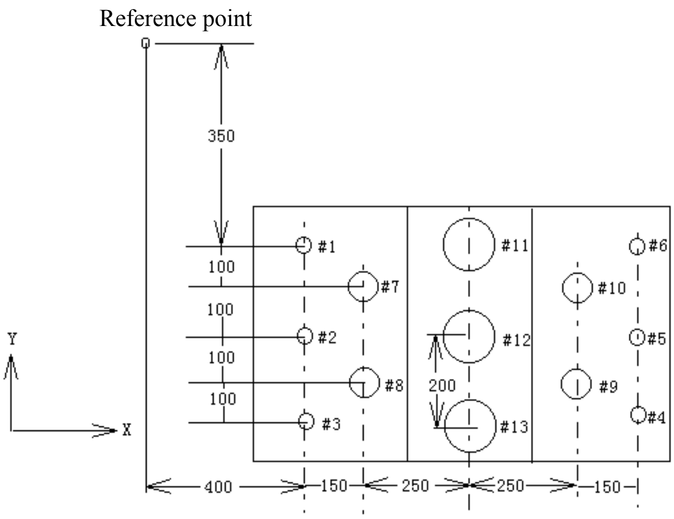
3.15.5 Examples for modal data specified in canned cycle

No.	Data Specification	Explanation
N0010	G00 X_ M3 ；	G00 positioning at the rapid traverse, and rotating the spindle;
N0020	G81 X_ Y_ Z_ R_ F_；	Because it is the beginning for the canned cycle, so the value needs to be specified for Z, R and F.
N0030	Y_；	The corresponding hole machining data is same to the previous hole, only the position Y is different, so G81Z_R_F_ can be omitted. As for the hole position is shifted for Y, hole machining is performed further by using the G81;
N0040	G82 X_ P_；	The hole position needs to be moved along the X axis as for the pervious one. The Z, R and F of previous hole and the P specified by this hole are taken as hole machining data by the G82;
N0050	G80 X_ Y_ M5 ；	The hole machining is not executed, all of the hole machining data are cancelled (except for the F); The GO positioning is performed with XY;
N0060	G85 X_ Z_ R_ P_；	The Z and R are needed to be specified newly because all of the data in previous block are cancelled, the above value specified is applied when the F is omitted. Although the P value is commanded, but it is not needed for this hole machining, so the P value is saved.
N0070	X_ Z_；	The Z is different compared with the previous hole, and the hole position just moves along the X axis;
N0080	G89 X_ Y_ D_；	The Z and R, P values separately specified by N0070 and N0060, the F value specified in N0020 are taken as hole machining data, which are used for G89 hole machining.
N0090	G112 I_ J_ F_ D_；	The fine-milling hole machined by G89 is performed by G112.
N0100	G0 X_ Y_ Z_；	positioning for a rectangle machining

N0110	G134 Z_R_I_J_K_U_D_；	Start machining the rectangle;
N0120	Y_I_J_K_U_D_；	Begins machining the second rectangle;
N0130	X_ Y_ I_J_K_U_D_；	Begins machining the 3rd rectangle;
N0140	G138 X_ Y_ R_ Z_ I_	The fine-milling inside the machined rectangle groove is to be performed, the corresponding data are needed;
	J_ K_ U_ D_ F_；	
N0150	G01 X_ Y_，	Cancel the hole machining mode and data (except for F); the G01 cutting feed is performed by XY.

Note: Address I, J, K and U of canned cycle G110, G111, G112, G113, G114, G115, G134, G135, G136, G137, G138 and G139 are not saved as canned cycle modal data, so the I, J and K values need to be specified in each block, or the alarm will be generated.

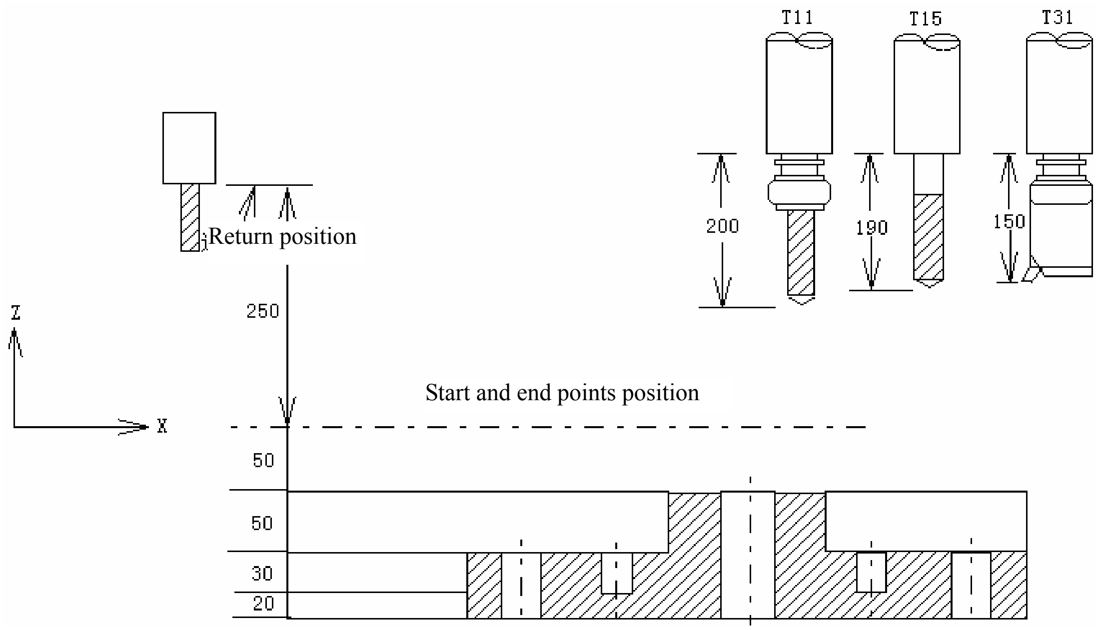
3.15.6 Examples for canned cycle and tool length compensation



The hole number from 1 to 6… drilling Φ10

The hole number from 7 to 10… drilling Φ20

The hole number from 11 to 13… boring Φ95 hole (depth is 50mm)



The values of offset numbers H11, H15 and H 31 are separately set to 200.0, 190.0 and 150.0, the program is as following:

N001 G92 X0 Y0 Z0 ；	The coordinate system is set at the reference point
N002 G90 G00 Z250.0 ；	
N003 G43 Z0 H11 ；	Plane tool length compensation is performed at the initial plane.
N004 S30 M3 ；	The spindle starts.
N005 G99 G81 X400.0 Y-350.0 ； Z-153.0 R-97.0 F120.0 ；	#1 hole is machined after positioning.
N006 Y-550.0 ；	#2 hole is machined after positioning, point R plane returned.
N007 G98 Y-750.0 ；	#3 hole is machined after positioning, initial plane returned.
N008 G99 X1200.0 ；	#4 hole is machined after positioning, point R plane returned.
N009 Y-550.0 ；	#5 hole is machined after positioning, point R plane returned.
N010 G98 Y-350.0 ；	#6 hole is machined after positioning, initial plane returned
N011 G00 X0 Y0 M5 ；	Reference point return, the spindle stops.
N012 G49 Z250.0 ；	Tool length compensation cancellation
N013 G43 Z0 H15 ；	Initial plane, tool length compensation.
N014 S20 M3 ；	Spindle starts
N015 G99 G82 X550.0 Y-450.0 ； Z-130.0 R-97.0 P30 F70 ；	#7 hole is machined after positioning, point R plane returned.
N016 G98 Y-650.0 ；	#8 hole is machined after positioning, initial plane returned.
N017 G99 X1050.0 ；	#9 hole is machined after positioning, point R plane returned.
N018 G98 Y-450.0 ；	#10 hole is machined after positioning, initial plane returned.
N019 G00 X0 Y0 M5 ；	Reference point return, the spindle stops.
N020 G49 Z250.0 ；	Tool length compensation cancellation.
N021 G43 Z0 H31 ；	Tool length compensation at initial plane.
N022 S10 M3 ；	Spindle starts.
N023 G85 G99 X800.0 Y-350.0 ； Z-153.0 R47.0 F50 ；	#11 hole is machined after positioning, point R plane returned.
N024 G91 Y-200.0 ； Y-200.0 ；	#12 and #13 are machined after positioning, point R plane returned.
N025 G00 G90 X0 Y0 M5 ；	Reference point return, the spindle stops.
N026 G49 Z0 ；	Tool length compensation cancellation
N027 M30 ；	Program stops.

3.16 Absolute and Incremental Commands G90 and G91

Format:

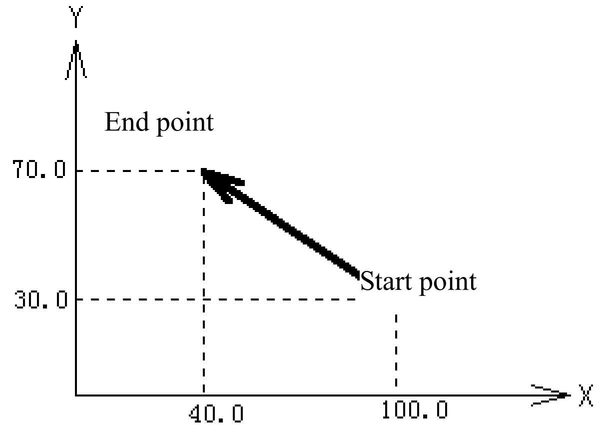
G90; Absolute command

G91; Incremental command

Function:

There are two kinds of modes for commanding axis offset, one is absolute command the other is incremental command. The absolute command is programmed by coordinate value of the terminal position by the axis movement. The incremental command is directly programmed by the movement value of the axis. They are separately specified by G90 and G91 commands.

Example:



The above movement is programmed by absolute and incremental commands, which is as follows:

G90 X40.0 Y70.0 ; or G91 X－60.0 Y40.0;

3.17 Workpiece Coordinate System Setting G92

Function: The workpiece coordinate system is set by setting the absolute coordinate in current position in the system (It is also called floating coordinate system). After the workpiece coordinate is set, the coordinate value is input in absolute programming in this coordinate system till the new workpiece coordinate system is set by G92.

Command explanation: G92, which is a non-modal G-command;

X: The new X axis absolute coordinate of current position;

Y: The new Y axis absolute coordinate of current position;

Z: The new Z axis absolute coordinate of current position;

Note: In G92 command, current coordinate value will be not changed if the X, Y and Z are not input, the program zero is set by the current coordinate value. When the X, Y or Z is not input, the coordinate axis not input keeps on the original set value.

3.18 Feed per min. G94, Feed per rev. G95

Format: G94 F<u>xxxx;</u> (F0001~ F8000, the leading zero can be omitted, the feedrate per min. is offered, mm/min.)

Function: The cutting feedrate is offered in mm/min unit when the G94 is modal G command. The G94 can be omitted if the current mode is G94.

Format: G95 Fxxxx; (F0.0001～F500, The leading zero can be omitted）

Command Function: The cutting feedrate is offered in mm/rev unit when the G95 is modal G command. The G95 can be omitted if the current mode is G95. The product of F command value (mm/r) and current spindle speed(r/min) is regarded as the command cutting feedrate to control the actual feedrate when the G95 Fxxxx is performed by system. The actual cutting feedrate varies with the spindle speed. The spindle cutting feed value per rev is specified by G95 Fxxxx, it can form even cutting grain on the surface of the workpiece. The machine should be installed spindle encoder when the G95 mode is used.

G94 and G95 are modal G commands in same group, one of them is effective in one time. G94 is initial modal G command, it is defaulted effective when the power is turned on.

The conversion formula for feed value per rev and per min is as following:

$$F_m = F_r \times S$$

Thereinto: F_m: Feed value per min (mm/min);

F_r: Feed value per rev per rev (mm/r);

S: Spindle speed (r/min).

The feedrate value is set by system data parameter No.030 when the power is turned on for the system; an F value is invariable after the F command is performed. The feedrate is 0 after the F0 is executed. The F value is invariable when the system is reset or emergency stop. **The feed override is memorized when the power is turned off.**

Related parameter:

System data parameter No.029: the exponential acceleration or deceleration time constant for cutting and manual feed;

System data parameter No.030: the lower value of exponential acceleration or deceleration on cutting feed;

System data parameter No.031: The upper limit value for cutting feedrate (X, Y and Z axes)

Note:

The cutting feedrate becomes uneven when the spindle speed is less than 1 rev/min in G95 mode; the actual feedrate has following error when the spindle speed fluctuates. In order to guarantee the machining quality, it is recommended that the spindle speed can not be lower than spindle servo or the lowest speed of effective torque introduced by inverter during machining.

3.19 G98, G99

Format:

G98；

G99；

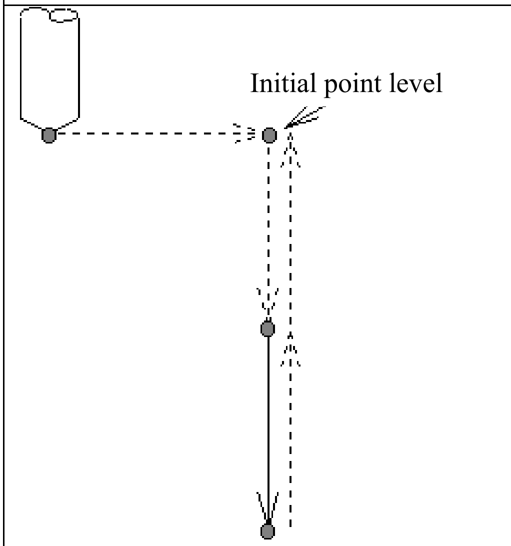
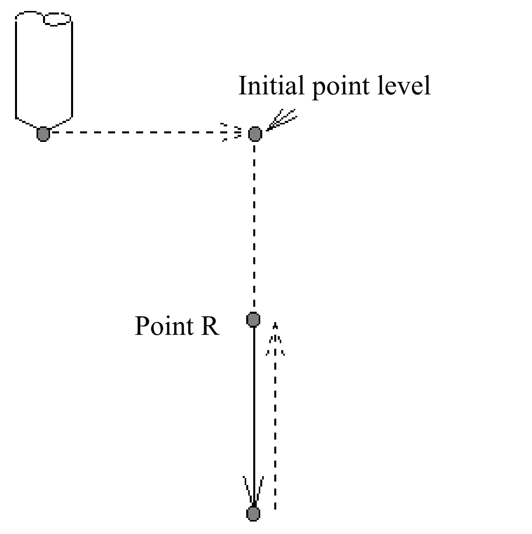
Function:

G98; Tool returns to the initial plane when the hole machining is returning.

G99; Tool returns to the point R plane when the hole machining is returning.

Explanation:

Modal G command

G98 (Return to initial plane)	G99 (Return to point R plane)
	

Refer to the explanation for canned cycle command.

3.20 Chamfering Function

A straight line or an arc is inserted into two figures; this is called Chamfering function. The tool can be smoothly transferred from one figure to another. GSK980MD owns two chamfering functions, one is linear chamfering, and the other is arc chamfering.

3.20.1 Linear chamfering

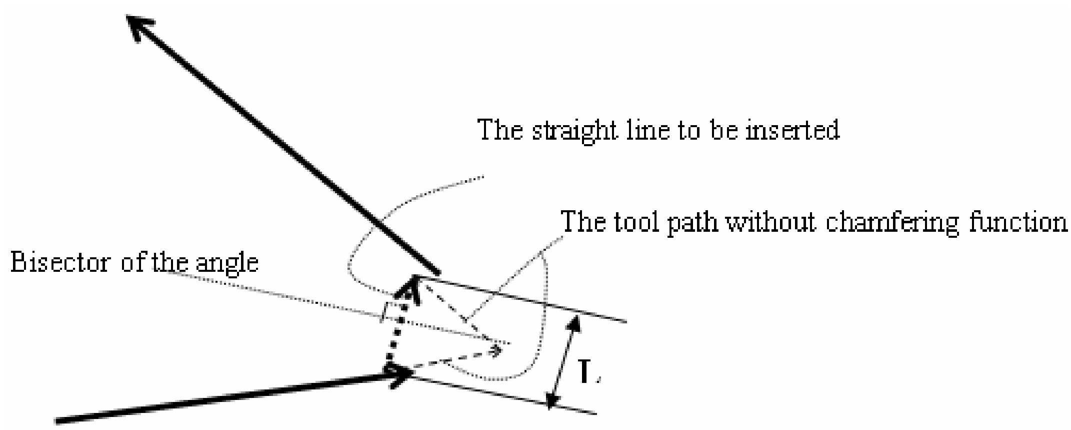
The linear chamfering is that a straight line is inserted between figures of the straight lines, the arcs, as well as the straight line and arc. The command address for linear chamfering is L. The data followed by command address L is the length of chamfering straight line. The linear chamfering should be employed in the G01, G02 or G03 command.

- **Linear to linear**

Format: G01 IP_ L_; (IP is axis movement command)

G01 IP_;

Function: A straight line is inserted into interpolation between 2 straight lines.

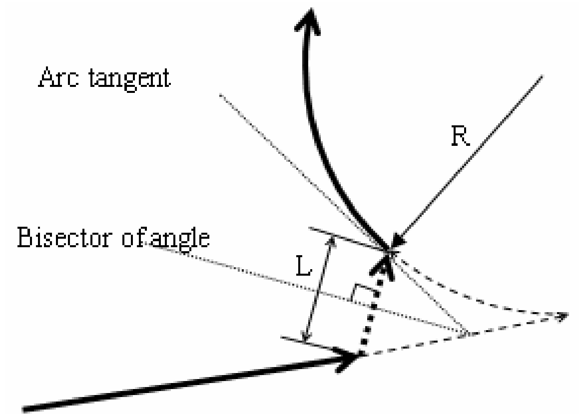


- **Linear to circular**

Format: G01 IP_ L_；

G02/G03 IP_ R_(I_ J_ K_)；

Function: A straight line is inserted between straight line and arc interpolation.



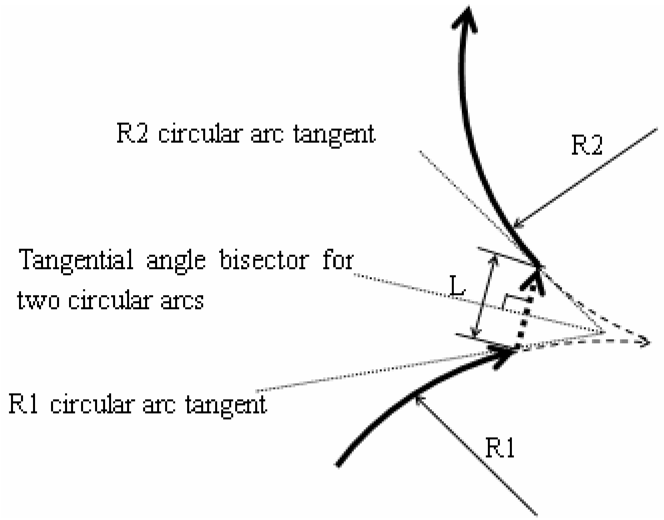
- **Circular to circular**

Format:

G02/G03 IP_ R_ (I_ J_ K_) L_；

G02/G03 IP_ R_(I_ J_ K_)；

Function: A straight line is inserted between two arc interpolations.

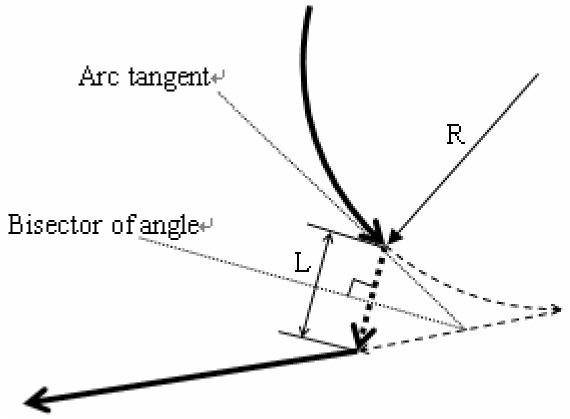


- **Circular to linear**

Format: G02/G03 IP_ R_(I_ J_ K_) L_；

G01 IP_；

Function: A straight line is inserted between the arc and linear interpolation.



3.20.2 Circular chamfering

An arc is inserted between the two linear figures, arc figures or linear and arc figures, this is called circular chamfering. Tangent transition is performed between arc and figure line. The command address is C for the arc chamfering, the data followed by command address C is the radius of chamfering arc. The arc chamfering should be employed in command G01, G02 or G03.

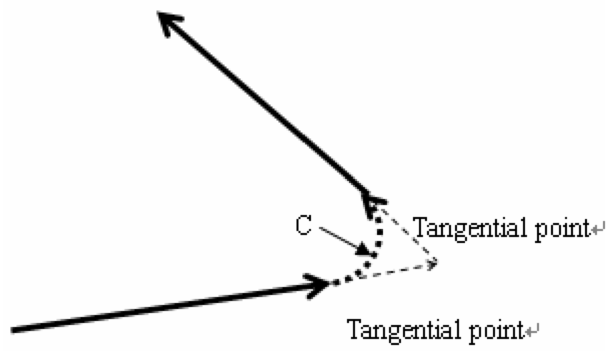
- **1. Linear to linear**

Format:

G01 IP_ C_；

G01 IP_；

Function: An arc is inserted between two linear interpolations, which it is tangential with two linear lines, the data followed by command address C is radius.



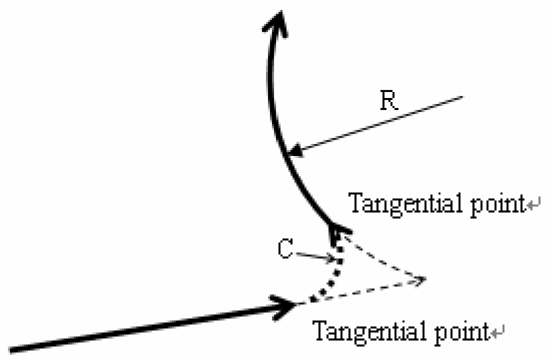
- **2. Linear to Circular**

Format:

G01 IP_ C_；

G02/G03 IP_ R_(I_ J_ K_) ；

Function: An arc is inserted at the intersection of straight line and arc, this arc is tangential with both the straight line and arc, the data followed by command address C is radius.



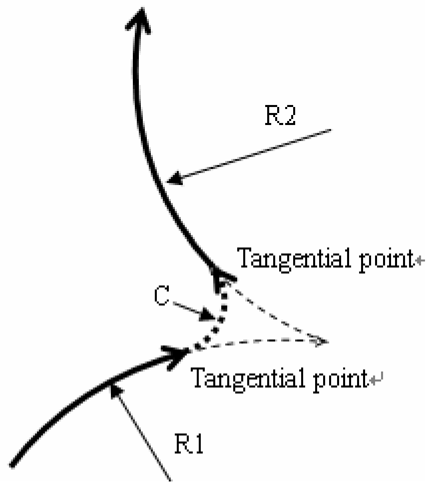
- **3. Circular to Circular**

Format:

G02/G03 IP_ R_(I_ J_ K_) C_;

G02/G03 IP_ R_(I_ J_ K_);

Function: An arc is inserted between two arc interpolations which it is tangential with two circulars, the data followed by the command address C is radius.



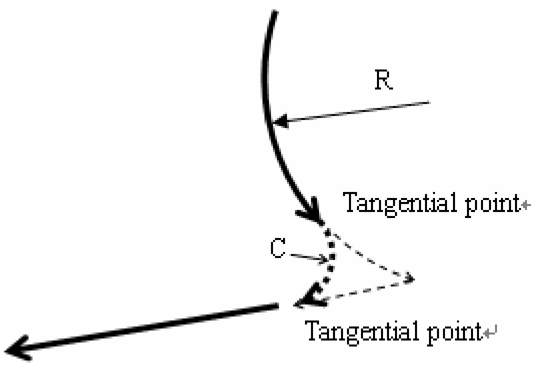
- **4. Circular to Linear**

Format:

G02/G03 IP_ R_(I_ J_ K_) C_;

G01 IP_;

Function: An arc is inserted at the intersection of arc and straight line, which is tangential with the
arc and straight line; the data following the command address C is radius.



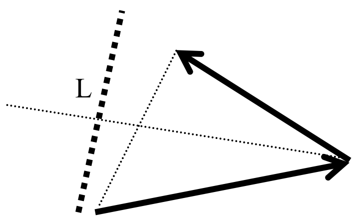
3.20.3 Exceptional Cases

The chamfering function is ineffective or alarm is issued in the following circumstances:

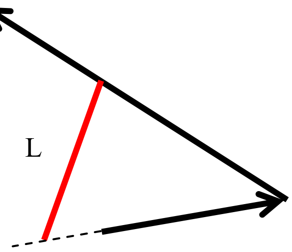
1. Linear chamfering

A. The chamfering function is ineffective when two interpolation lines is shown on the same line.

B. If the chamfering linear length is too long, and the CNC alarm occurs.



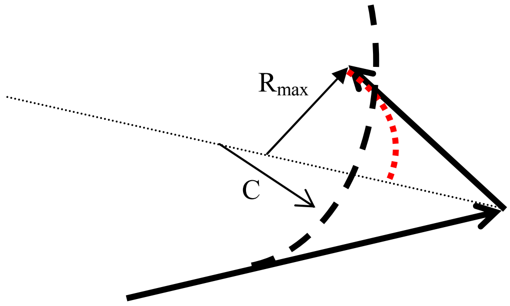
C. If some line (arc) is too short, the alarm occurs.



2. Arc Chamfering

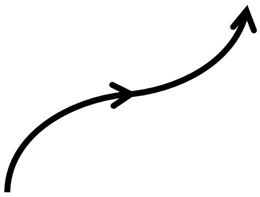
A. The arc chamfering function is disabled when two interpolation lines are shown on the same line.

B. If the chamfering radius is excessive, the CNC alarm occurs.



C. The arc chamfering function is disabled when the line is tangential with arc or the arc is tangential with line.

D. The arc chamfering function is disabled when the arcs are tangent.



Note 1: The chamfering function can be performed only in the plane specified by G17, G18 or G19, these functions can not be performed in parallel axes.
Note 2: Changing the coordinate system by G92 or G54 to G59, or, the block followed by performing the reference point return from G28 to G30 can not specify the chamfering.
Note 3: Chamfering function can not be employed in the DNC mode.

3.21 Rigid Tapping

The right-handed tapping cycle (G84) and left-handed tapping cycle (G74) may be performed in standard mode or rigid tapping mode. In standard mode, the spindle is rotated and stopped along with a movement along the tapping axis using miscellaneous functions M03 (rotating the spindle cloclwise), M04 (rotating the spindle counterclockwise), and M05 (stopping the spindle) to perform tapping.

In rigid mode, tapping is performed by controlling the spindle motor as if it were a servo motor and by interpolating between the tapping axis and spindle. When tapping is performed in rigid mode, the spindle rotates one turn every time a certain feed (thread lead) which takes place along the tapping axis. This operation does not vary even during accleration or deceleration.

3.21.1 Rigid Tapping

Code format:

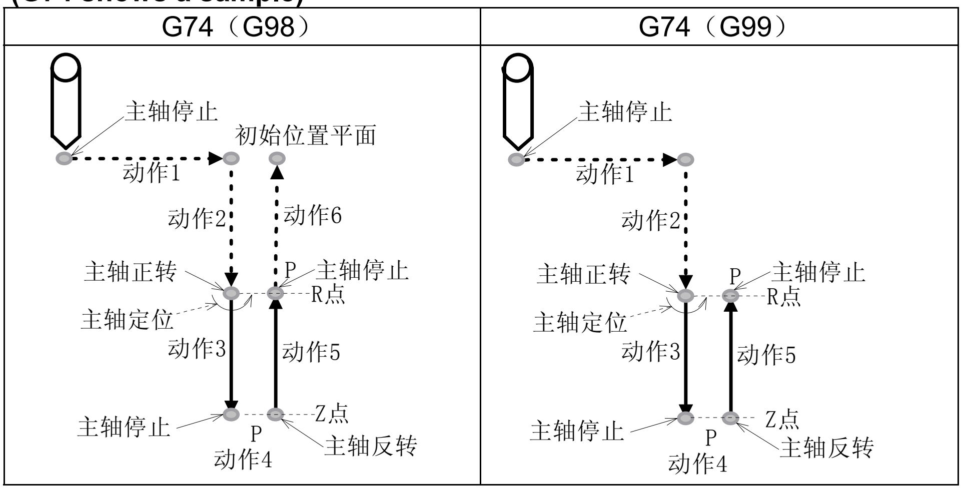
Left-handed rigid tapping: G74 X_ Y_ Z_ R_ P_ F（I）_ L_ C_

Right-handed rigid tapping: G84 X_ Y_ Z_ R_ P_ F（I）_ L_ C_

Code function: In rigid mode, tapping is performed by controlling the spindle motor as if it were a servo motor and by interpolating between the tapping axis and spindle. When tapping is performed in rigid mode, the spindle rotates one turn every time a certain feed (thread lead) which takes place along the tapping axis. This operation does not vary even during accleration or deceleration.

Cycle process: (1) Position to the XY plane at the rapid traverse rate;
(2) Reduce to the point R plane rapidly, then to the position where the C is specified at the rapid traverse rate;
(3) Tapping is performed to the bottom of the hole, then the spindle stops;
(4) Dwell time P is performed if the P is specified;
(5) Spindle rotates reversely returns to the point R plane, the spindle then stops; dwell time P is performed if the P is specified;
(6) Return to the origin plane if the command is G98;

Code path: (G74 shows a sample)



Explanations:

When the tapping operation 3 is being performed, the feedrate override can not be adjusted; when the operation 5 is perfoming, the speed override value is set by the data parameter 084, when the data parameter 084 is set to 0, the override value is fixed as 100%

When the tapping operation 3 is being performed, the linear acceleration or deceleration constant value is set by the data parameter 082; when the tapping operation 5 is performed, the linear acceleration constant value is set by data parameter 083, if the data parameter 083 is se to 0, the linear acceleration/deceleration time constant in operation 5 is set by the data parameter 082.

3.21.2 Peck Rigid Tapping

Code format:

(High-speed/standard) peck left-handed rigid tapping: G74 X_ Y_ Z_ R_ P_ F（I）_ L_ Q_ C_
(High-speed/standard) peck right-handed rigid tapping: G84 X_ Y_ Z_ R_ P_ F（I）_ L_ Q_ C_

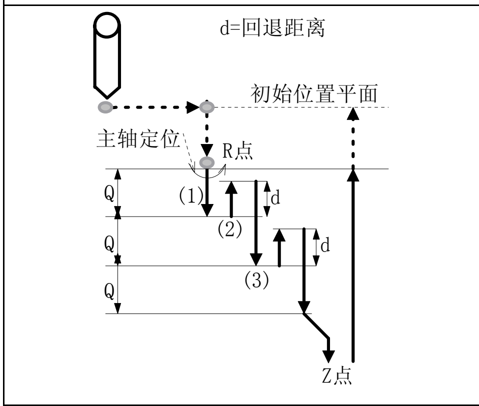
Code function: When the peck tapping is performed in rigid tapping, due to chips sticking to the tool or increased cutting resistance, in such cases, the preferable tapping can be performed by the peck rigid tapping.

High-speed peck rigid tapping:

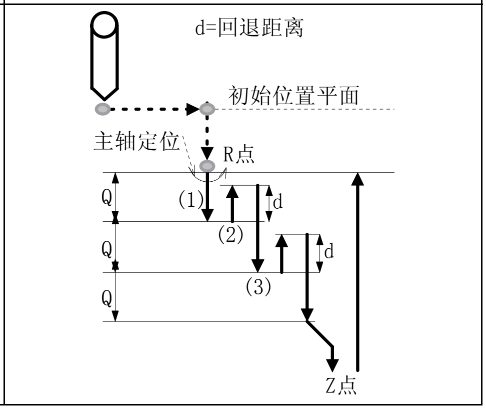
When the RTPCP of state parameter No.025 is set to 1, the high-speed peck rigid tapping cycle is selected.

After positioning along the X- and Y-axes, rapid traverse is performed to point R, then position to the place where specifies by C. From point R, cutting is performed with depth Q (depth of cut for each cutting feed), then the tool is retracted by distance d, the retraction speed can be overridden. When point Z has been reached, the spindle is stopped, and then rotated in the reverse direction for retraction. The tool retracts to the point R, the spindle stops. If it is G98 state, rapidly move to the initial position, the Figure is shown below:

G74、G84（G98）



G74、G84（G99）

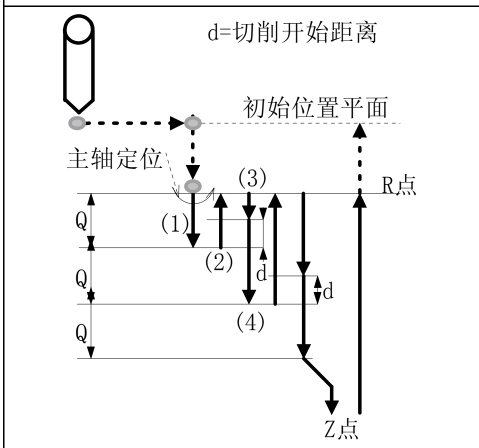


Standard peck rigid tapping:

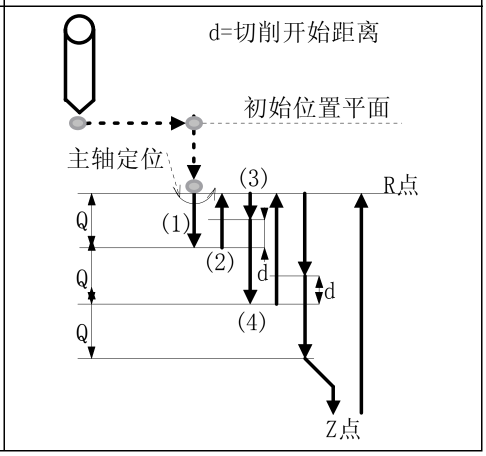
When the RTPCP of state parameter No.025 is set to 1, the standard peck rigid tapping cycle is selected.

After positioning along the X- and Y-axes, rapid traverse is performed to point R, then position to the place where specifies by C. From point R, cutting is performed with depth Q (depth of cut for each cutting feed), then the tool is retracted by distance d, the retraction speed can be overridden. The position is performed from point R to a distance d from the end of the last cutting, which is where cutting is restarted, and the cutting feed is performed. When point Z has been reached, the spindle is stopped, then rotated in the reverse direction for retraction. The tool retracts to the point R, the spindle stops. If it is G98 state, rapidly move to the initial position, the Figure is shown below:

G74、G84（G98）



G74、G84（G99）



Explanations:

When tapping feed is performing, the speed override can not be adjusted; when the retraction is

performed, the speed override value is set by data parameter 084, when the data parameter 084 is set to 0, the override value is fixed as 100%.

The linear acceleration or deceleration constant value in tapping feed is set by data parameter 082, the linear acceleration or deceleration constant in retraction is set by data parameter 083, if the 083 is set to 0, the acceleration or deceleration constant in retaction is then set by data parameter 082. The start speed both tapping feed and retraction are set by data parameter 081, and the retraction distance d is set by data parameter 085.

3.21.3 Address Explanation

Specified content	Address	Command address explanation
Hole position data	X、Y	Specify the hole position by the absolute value or incremental
Aparture machining data	R	From the initial plane to the point distance
	Z	Depth of a hole, the distance from point R to the bottom of the hole
	P	Specify the dwell time at the bottom of the hole or at point R when a return is made. The dwell does not perform when it is not input or the value is 0.
	Q	Tool infeed value of peck tapping
	L	It indicates that the consecutive maching cycle of L holes are performed on this line segment from start (the start position of block) to XY coordinate position. The continued drilling may not perform if it is not input or the value is 0.
	F	Metric thread leading, the solution range: 0.001~500mm. The alarm 201 may alarm if it is not input.
	I	The number of the thread head per/inch, the solution range is 0.06~25400 gear/inch
	C	Start angle

3.21.4 Technic Specification

- Acceleration/deceleration

  Rigid tapping adopts the acceleration or deceleration before a straight line to control.
- Override

  The override regulation is invalid for rigid tapping infeed, but the override value can be adjusted or not which is determined by data parameter.
- Dry run

  G84/G74 can be used a dry run, the dry run equals to the feedrate along Z axis. The override adjustment is invalid in dry run.
- Machine lock

  G84/G74 can be used a machine lock, the tapping axis and spindle axis are not moved when the machine lock is enabled.
- Resetting

  The resetting can be reset the tapping when the rigid tapping is performed, but the G74/G84 can be not be reset.
- Dwell

  The dwell is disabled.
- Working

  G84/G74 is only valid in Auto or MDI mdoe.

- Manual feed
  The rigid tapping can not used for manual feed.
- Tool length compensation
  If the tool length compensation (G43, G44 or G49) is specified in canned cycle, the offset value is added till position to the point R.
- Cutter compensation
  Cutter compensation is ignored in canned cycle.
- Axis switching
  The Z axis tapping can only be performed in rigid mode.
- S code
  If the command speed is more than the maximum speed, the alarm may occur.
- M29
  Specify an axis movement code between M29 and G84/G74 causes alarm.
- P/Q

If they are specified in non-drilling block (If they are specified in a block that does not perform drilling), they are not stored as modal data. When Q0 is specified, the peck rigid tapping cycle is not performed.

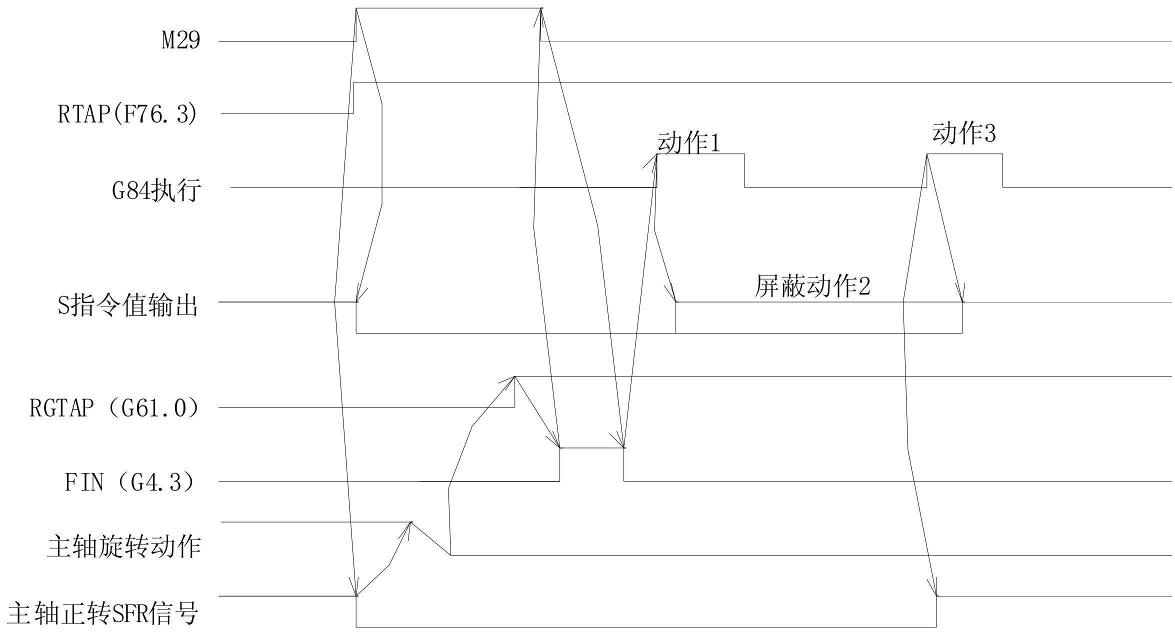
Specify them in tapping block, they are stored as modal data, when the tapping command is retracted, either Q modal (did it).

- Cancellation
  Do not specify a group 01 G code and G84/G74 in the same block.
- A Cs contour control is used with rigid tapping at the same time.

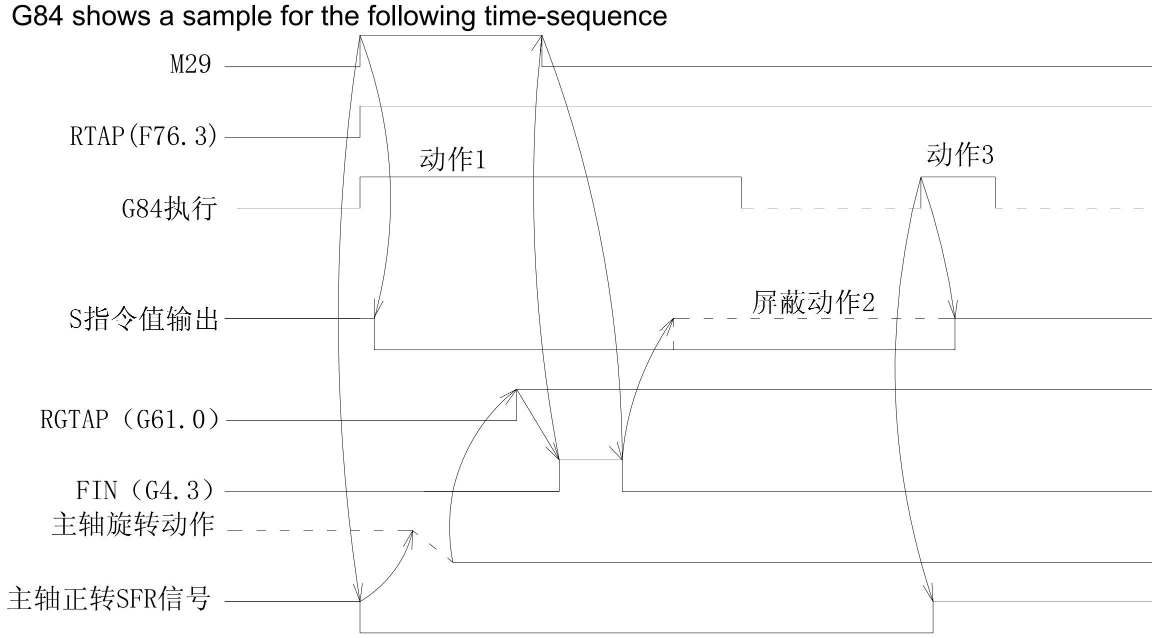
CS axis selects a speed mode or position mode which is determined by CON (G27.7), but, the system is rigid tapping mode, regardless of the value of CON. After the rigid tapping is cancelled, the rotation axis is either CS axis or common one which is determined by state parameter. The C axis can not be moved in manual mode when the rigid tapping is not cancelled.

3.21.5 Specify a Rigid Tapping Mode

- Specify M29 before G74/G84
  G84 shows a sample for the following time-sequence



- Specify M29 and G74/G84 at the same block
  G84 shows a sample for the following time-sequence



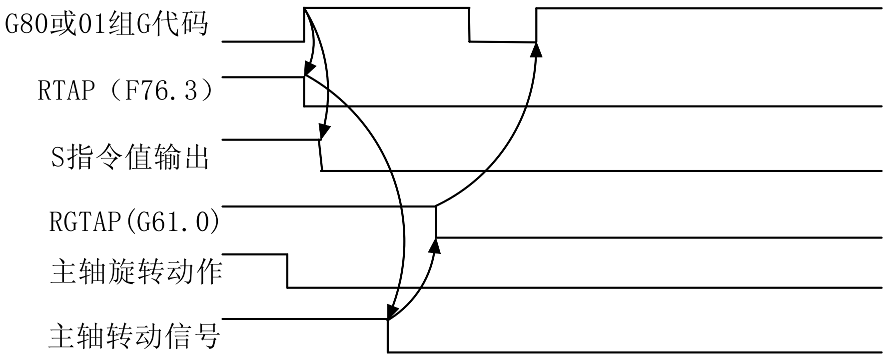
- The explanation of time sequence
  The spindle rotation operation means that the rotation axis is shifted to the position control mode (namaly, the servo spindle is needed to send a switch signal in position mode), and check the position mode arrial signal of servo spindle.

3.21.6 The cancellation of rigid tapping mode

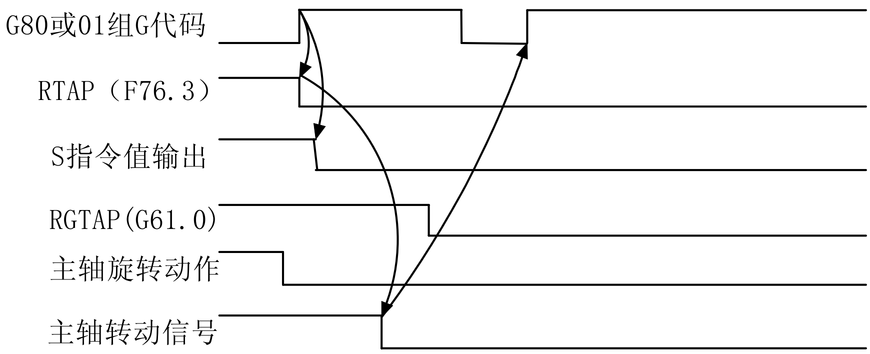
- The rigid tapping mode is canceled by G80
- Specify other canned cycles by G codes
- The other G codes of group 1.
- CNC resetting

The signal descending of F76.3 along the signal with canceling the rigid tapping of PLC, if the state RTCRG of parameter 025 is equal to 1, the system is then performed the next block without waiting for the rigid tapping mode signal which G61.0 is set to 0;

When the state parameter 025.2 (CRG) =0, the time sequence is as follows:



When the state parameter 025.2 (CRG) =1, the time sequence is as follows:



3.21.7 F and G Signals

RGTAP (G61.0): Rigid tapping signal

When the M 29 is commanded, PMC enters the rigid tapping mode, and the signal is then set to 1 to inform the CNC

1: PMC enters the rigid tapping mode

0: PMC does not enter the rigid tapping mode

If this signal does not set to 1, after the M29 has been commanded, the alarm may occur in the block of G74/G84.

RGSPM, RGSPP (F65.1, 0) spindle turning signal

When the rigid tapping is performed, the signal is informed to the PMC whether the current spindle is CCW (positive) or CW (negative).

RGSPM: 1 spindle CW (negative)   RGSPP: 1 spindle CCW (positive)

In rigid tapping, these signals are output when the spindle is rotated. In the mode of rigid tapping, when the spindle is positioned at the hole or stoppted at the bottom of the hole or R position, these signals are not output.

In the mode of rigid tapping, when the spindle is positioned at the inter-locked stop, machine lock or Z axis ignorance states, the spindle does not regard as a stop state, in this case, these signals are output. These signals are only enabled in rigid tapping, and they are all set to 0 in the normal spindle control mode.

RTAP (F76.3): Rigid tapping process signal

This signal informs PMC which has been in the mode of rigid tapping or not. The CNC is in the mode of rigid tapping currently when the signal is set to 1.

This signal can be locked M29, PLC has been commanded the rigid tapping mode, the PMC is then treated with the correspinding logic, and this signal can be replaced the lock of M29, even so, the FIN singl of M29 is not ignored still.

3.21.8 Alarm Message

Alarm No.	Display Content	Explanation
218	Fail to specify the tool pitch F value in G74 or G84	Fail to specify F value
230	The spindle feed can not be performed due to the S value is 0.	S value is 0, or S code does not specify.
231	S value exceeds the maximum spindle speed allowed with rigid tapping	S value exceeds the setting value of data parameter 086
232	Other axis movement codes are specified between M29 and G74/G84.	Specify a axis movement between M29 and G74/G84__
233	G61.0 signal is abnormal in rigid tapping mode	Rigid tapping signal G61.0 is not 1 during performing in G74/G84.
234	Specify M29 repeatedly	Specify M29 or it is consecutively specified more than twice in rigid tapping.

3.21.9 Program Example

G84 shows an example for the following program

```
O1000 (Rigid tapping example);
G0 X0 Y0 Z0；
M29 S200;
G84 X10 Y10 Z-10 R-5 P2000 F2 C20;
X20 C40
G80;
M30;
```

CHAPTER 4 CONTROL FUNCTION of ADDITIONAL AXIS

4.1 General

The additional axis is determined by the struction design of the machine, sometimes, an additional axis is required, for example, the cycle working table, rotation working table. This axis can be designed as both a linear axis and rotation axis. The basis controllable number of 980MDa is three axes, the maximum axis is 5-axis (Cs axis included). Namely, two additional axes are added based upon the original one —— the 4^{th} and the 5^{th} axes, in this case, the relative functions of additional linear axis and rotation axis can be performed.

4.2 Axis Name

The names of three basis axes are always X, Y or Z. The axis name of additional axis can be set to A, B or C using data parameter No.202 and No.203.

- **Default axis name**

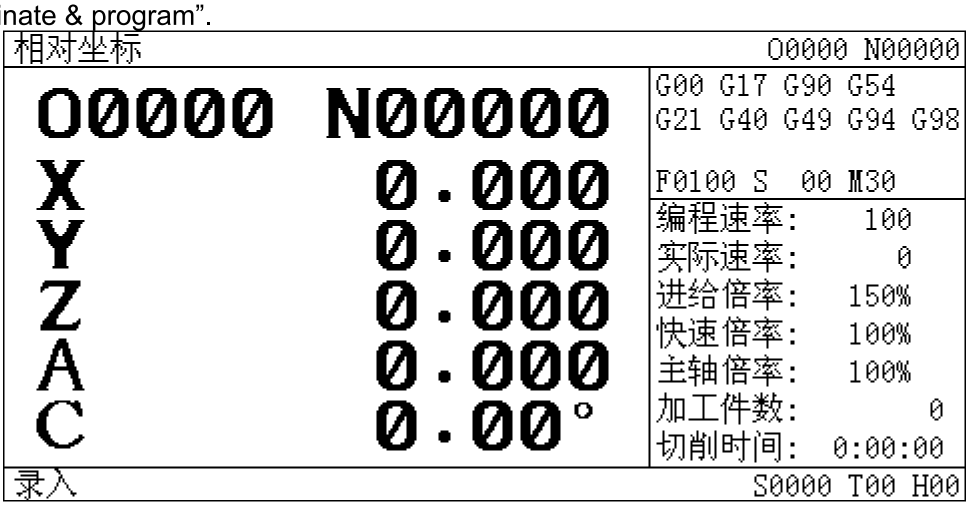
When the axis name does not set, the axis name of the 4^{th} one is an additional axis by default; the axis name of the 5^{th} one is C.

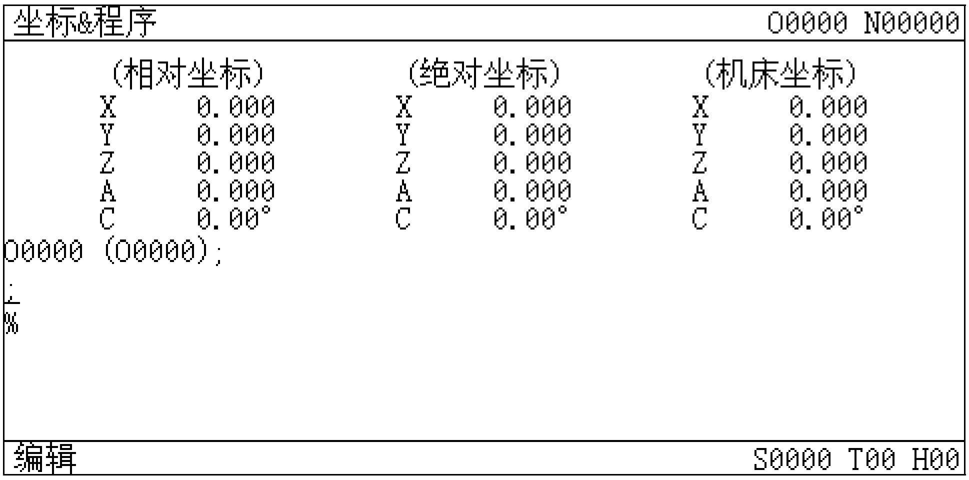
- **Repeated axis name**

When the axis name is same between the added 4^{th} axis and the 5^{th} axis, P/S alarm may issue.

4.3 Axis Display

When the additional axis is treated as rotation axis, the least incremental of the rotation axis is 0.01° (degree), so the 3^{rd} digit of the decimal is displayed in unit. If it is set to a linear axis, the display is same as the basis three axes (X, Y or Z). When the 4^{th} axis is set to a linear axis, the 5^{th} is set to a rotation axis, the axis is displayed at the interface of "related coordinate" and "coordinate & program".





4.4 Axis Startup

The Bit 1 (ROSx) of data parameter No.026 and Bit0 (ROTx) of data parameter No.028 are separately set to use whether the 4th axis and the 5th axis is either the linear axis or rotation axis. The parameter settings are shown below:

ROS	ROT	Content
0	0	Linear axis 1. It can be switched between metric and inch; 2. All of the coordinate values are linear axis; 3. The stored pitch error compeneation is linear axis.
0	1	Rotation axis (Type A) 1. It can not be switched between metric and inch; 2. The machine coordinates are cycled based on the setting value of data parameter No.189/No.190. Whether the absolute coordinate and relative coordinate are cycled which based upon the data parameter No.027/No.029; 3. The stored pitch error compensation is rotation axis; 4. The movement amount is less than one turn when the reference position (G28, G30) is returned.
1	0	Ineffective setting (forbidden)
1	1	Rotation axis (Type B) 1. It can not be switched between metric and inch; 2. The machine coordinate is linear axis; whether the absolute coordinate and relative coordinate are cycled which based on the data parameter No.027/No.029. 3. The stored pitch error compeneation is linear axis.

Note: The start of the function of the Cs axis, the Bit 5 digits (RCSx) of the state parameter No.026 or No.028 can be set whether the function of Cs axis is enabled when the rotation axis is enabled (ROTx=1).

4.5 The Additional Axis is Linear Axis

When the additional axes (the 4th and the 5th axes) are set to linear axes, its functions are same as the basis three axes.

- **Realizable operation**

  1. Rapid traverse (Positioning): G90/91 G00 X_ Y_ Z_ A_;

  2. Cutting feed: G90/91 G01 X_ Y_ Z_ A_ F_;

  3. Skip function: G90/91 G31 X_ Y_ Z_ A_ F_;

4. Reference position return: G28/29/30 X_ Y_ Z_ A_ F_;
5. G92 coordinate setting: G92 X_ Y_ Z_ A_ ;
6. Manual/Step/MPG feed, Manual machine zero return.

Note: When there is no special explanation in the subsequent narration, the axis names of additional linear axes are expressed with "A".

- **Explanations**

1. When the additional linear axis rapidly moves or performs, it can be simultaneously specified with any axes of X, Y and Z. Each axis may rapidly move at its customized speed.

2. When the additonal linear axis is performed the cutting feed (G01) or used a skip function (G31), it can be simultaneously specified with any axes of X, Y and Z. in this case, the linear axis does not has an individual feedrate F but depend on each axis specified at a same time, which it is started or ended together with the specified each axis; namaly, the additional axis is shared with the basis three-axis linkage.

3. The additional linear axis can not performed a circular arc cutting (G02/03), otherwise, the P/S alarm may occur.

4. The pitch error of additional linear axis and the compensation function of inverse interval are same as the basis three-axis.

4.6 The additional axis is rotation axis

- **Input unit**

The pulse equivlance (namally, the least input unit) of 980MDa rotation axis is 0.01° (degree); the maximum vlaue of output pulse frequence is 500K.

When the selection is output based on the direction of pulse adding, it can be inputted a maximum speed n=60*f/36000=833.33 (rev./min.)

- **Rotation axis speed**

The feedrate of rotation axis is regarded the degree/min. as a unit. When the linear axis X, Y and Z is performed a linear interporlation with the rotation axis, the speed specified with F (mm/min) is the compound feedrate both X, Y and Z and the rotation axis.

Feedrate calculation: Calculate the required time when the feedrate is performed to the end; then, the feedrate unit of rotation axis is changed into degree/min..

For example: G91 G01 X20.0 C40.0 F300.0;

The unit of C axis is switched into 40mm from the 40.0 degree. The required time to the end is:

$$\frac{\sqrt{20^2+40^2}}{300}=0.14907 \quad \text{(min.)}$$

The speed of C axis is:

$$\frac{40}{0.14907}=0268.3 \quad \text{(degree/min.)}$$

Note: When there is no special explanation in the subsequent narration, the axis names of additional linear axes are expressed with "C".

- **The cycle function of rotation axis**

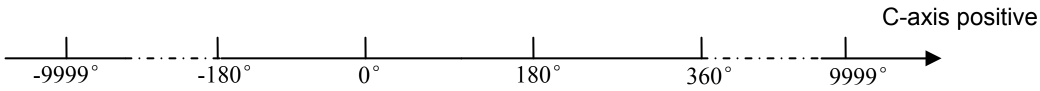
The coordinate cycle function of the additional rotation axis setting is enabled, which can be

avoided the coordinate value is overflowed from the rotation axis; the coordinate value will be cycled based on the setting value of data parameter No.189/No.190 (the movement amount of each axis for the rotation axis).

When the coordinate cycle function of the additional rotation axis setting is disabled, the coordinate value may change based on the linear axis, the programming command is also same to the one of the linear axis;

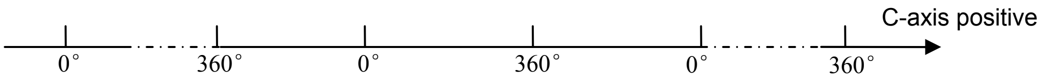
Two kinds of coordinates change are shown below:

(1) When the coordinate cycle is disabled:



The above-mentioned may occur: 1. The machine coordinate value of rotation axis (Type B)
2. The absolute coordinate value in data parameter No.027 ROAx=0 (absolute coordinate cycle function is disabled)
3. The relative coordinate value in data parameter No.027 RRLx=0 (relative coordinate cycle function is disabled)

(2) When the coordinate cycle is enabled:



The above-mentioned may occur: 1. The machine coordinate value of rotation axis (Type A)
2. The absolute coordinate value in data parameter No.027 ROAx=1 (absolute coordinate cycle function is enabled)
3. The relative coordinate value in data parameter No.027 RRLx=1 (relative coordinate cycle function is enabled)

Note 1: Refer to the Section of "Installation and connection" of the *Parameter Explanation of Chapter Three* for the parameter setting of additional rotation axis.

Note 2: When there is no special explanation in the subsequent narration, the movement amount of each revolution of the additional rotation axis is expressed with 360°.

- **The pitch error compensation function of rotation axis**

When the additional axis is a linear axis or rotation axis (Type B), the pitch error compensation mode is same as the common linear axis. The pitch error compensation function is performed when the additional axis is regarded as rotation axis (Type A), refer to the following examples:

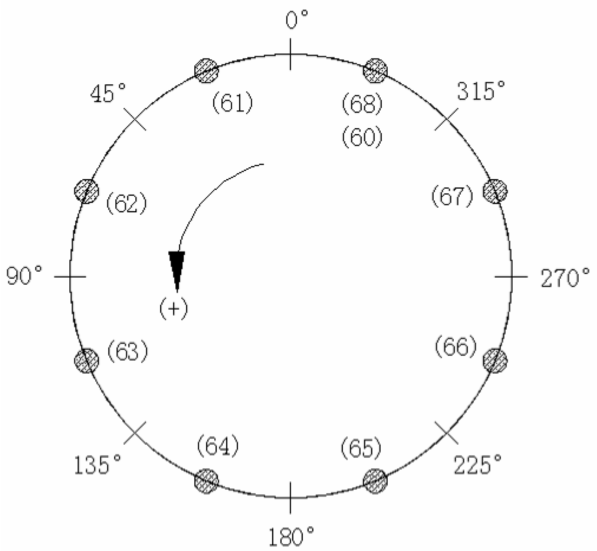
- Movement amount per revolution: 360°
- Pitch error pisition interval: 45°
- The compensation position number of reference position: 60

After the above parameters are set, the farthest compensation position number along the negative rotation axis which equals to the compensation position number of reference position;

The farthest compensation number along positive direction is shown below:

The compensation position number of reference point + (movement amount per revolution/compensation position interval) = 60 + 360/45 = 68;

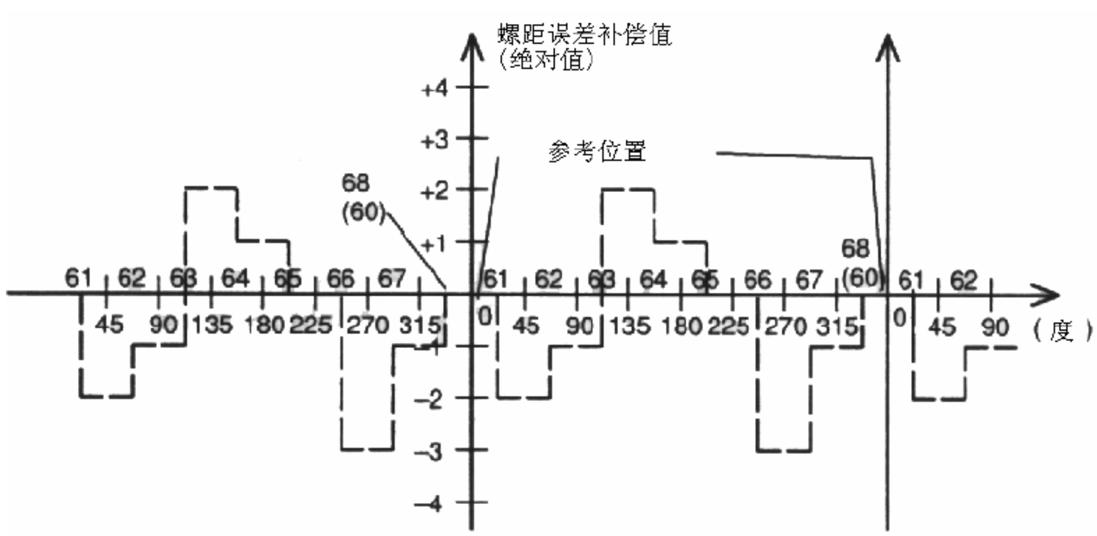
The corresponding relationships between machine coordinate and compensation position number are as follows:



The position error may occur if the total of compensation value from position 61~68 is not 0; there is not alternative other than to set a same value at the compensation position both 60 and 68. (Because the 60 and 68 are shared a same position at the circle);

The compensation sample is shown below:

NO.	60	61	62	63	64	65	66	67	68
Compensation value	1	-2	1	3	-1	-1	-3	2	1



- **The reverse interval compensation function of rotation axis**

The reverse interval compensation never changes regardless of the linear axis or rotation axis; however, the compensation unit of the rotation axis is 0.01° (deg), and the linear axis is 0.001 (mm);

4.7 The zero return D of rotation axis

The selection axis has four zero return methods: zero return method A, B, C and D. Wherein, the zero return methods A, B and C are same as the one of the linear axis. Only the D is a special zero return method for the rotation axis.

- **Setting of the zero return method D**

The method D is only valid to the rotation axis.

Zero return can be performed for this rotation axis using the mode D after the 4th and the 5th axes are set to rotation axes based on the Bit6 of data parameter No.027 and No.029 are set to 1.

If the 4th and 5th axes are disabled or linear axes, then the Bit6 of state parameter No.027 and No.029 are invalid.

0 2 7		RRT4						

RRT4 = 1: The zero return mode of the 4th rotation axis is used the mode D;

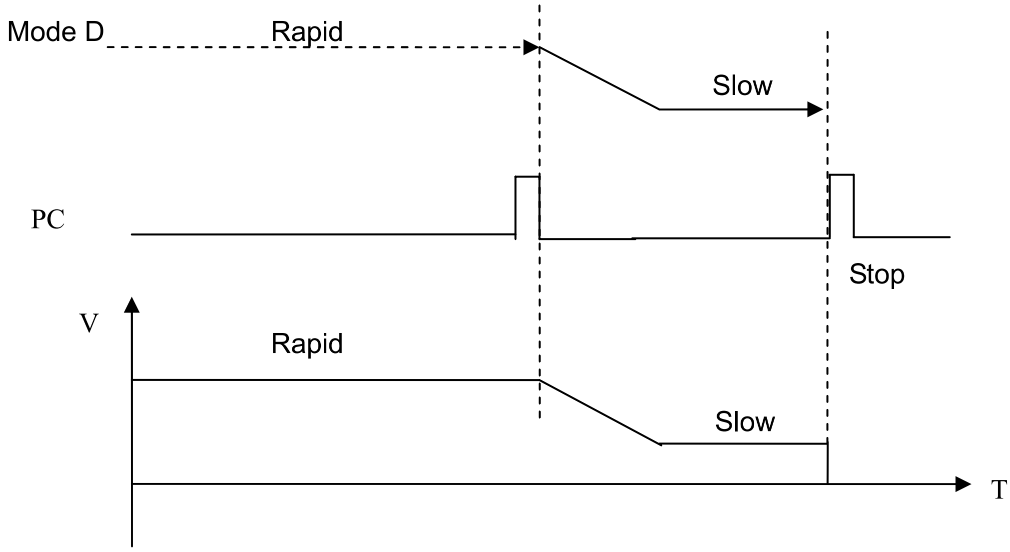
= 0: The zero return mode of the 4th rotation axis is used the mode A, B, and C.

0 2 9		RRT5						

RRT5 = 1: The zero return mode of the 5th rotation axis is used the mode D;

= 0: The zero return mode of the 5th rotation axis is used the mode A, B and C.

- **The time sequence and process of the zero return mode D**



The process of zero return

1. Select the machine zero return mode and press the manual positive feed key, the corresponding axis moves toward the zero point at the rapid traverse rate.

2. When the one-turn signal (PC) of servo axis is carried out, the system is decelerated to the zero return low speed, in this case, check the trailing edge of PC signal.

3. The system continuously and forward operates in the zero return low speed.

4. When the system meets one-turn signal (PC) of servo axis again, the movement stops, simultaneously, the corresponding indicator of zero return end on operator panel goes on. The machine zero return operation ends. In this case, check the rising edge of PC signal.

4.8 The Function of Cs Axis

General

The spindle is treated as the servo feed axis to rotate and position by the position movement command. Run speed is: degree/min., it can be interpolated together with other feed axes to machine a contour curve.

Increment system: the least input increment: 0.01deg

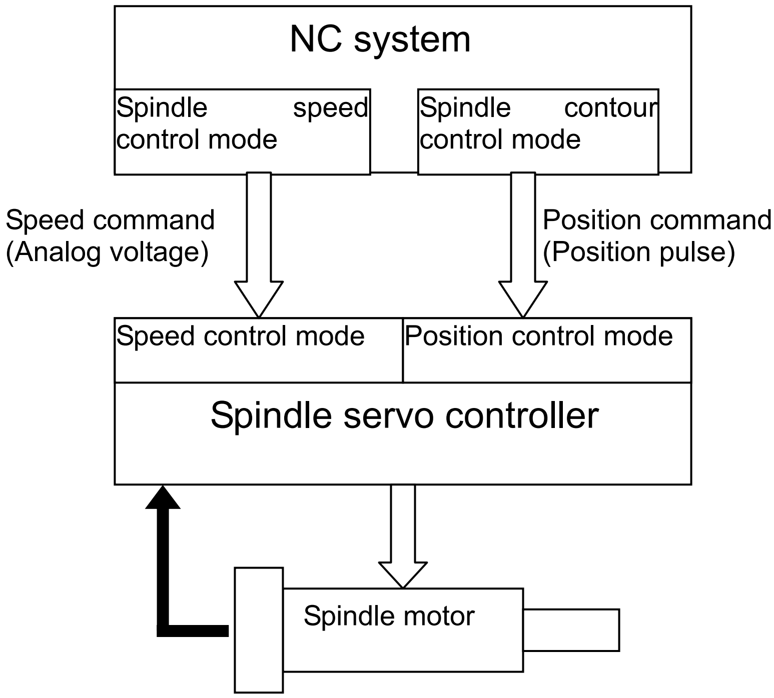
The least command increment: 0.01deg

Explanation: NC has two control modes for the spindle.

- Spindle speed control mode. The spindle speed can be controlled by the speed command (Namely, analog voltage).
- Spindle contour control mode (It is also called CS contour control). The spindle position can be controlled by the position command (Namely, position pulse).

So, NC is required the spindle servo control unit has two control modes for the control of the spindle motor

- When NC is at the speed control mode for the control of the spindle, the spindle servo control unit can receive a speed command issued from NC to control the rotation speed of spindle motor.
- When NC is at the contour control mode for the control of the spindle, the spindle servo drive unit also can receive a position command issued from NC to control the motor operates to a specified position.



Set Cs contour control axis

In the 980MDa system, only the additional axis (the 4th or the 5th axis) can be set to a Cs contour control axis. But, two Cs axes can not be set at the same time. Before the Cs axis setting is valid, this axis must be set to a rotation axis. Otherwise, Cs axis setting is invalid.

0	2	6		***	***	RCS4	***	***	***	ROS4	ROT4

RCS4 =1: The CS axis function of the 4th axis is enabled;

=0: The CS axis function of the 4th axis is disabled.

ROS4, ROT4: Set the type of the 4th axis;

	Linear axis	Type A rotation axis	Type B rotation axis	Invalid
ROT4	0	1	1	0
ROS4	0	0	1	1

0	2	8		***	***	RCS5	***	***	***	ROS5	ROT5

RCS5 =1: The CS axis function of the 5th axis is enabled.

=0: The CS axis function of the 5th axis is disabled.

ROS5, ROT5: Set the type of the 5th axis;

	Linear axis	Type A rotation axis	Type B rotation axis	Invalid
ROT5	0	1	1	0
ROS5	0	0	1	1

The switch between spindle speed control and CS contour control

The NC switching of spindle control mode is performed by the CON signal of PLC.

In the CS contour control mode of NC, the CS contour control axis, as the common servo axis, can be performed manually or automatically.

- From spindle speed control shifts to the Cs contour control

  Set the CON (G027#7) to 1, then the spindle can be set in the Cs contour control mode. If the switch is performed during the spindle rotation, the spindle is immediately stopped and then shifts.

- From Cs contour control shifts to the spindle speed control

  Set the CON (G027#7) to 0, the spindle is then set in the spindle speed control mode. Confirm the spindle movement command has been ended before shifting, if the shift is performed when the spindle is being moved, the system will alarm.

The reference position return of Cs contour control axis

After the spindle is shifted to the Cs contour control mode from the speed control mode, the current position is not confirmed, the spindle should be returned to the reference position.

The reference position return of Cs contour control axis is as follows:

- Manual reference position return

  After the spindle enters the Cs contour control mode, shift to the machine zero return mode. The zero return of Cs axis is performed opening the feed axis and the direction selection signal ＋Jn (G100) or －Jn (G102).

- Automatic

  Specify G28 after the spindle enters the Cs contour control mode, and the spindle moves to the intermediate point and then return to the reference position.

ZPn (F094) becomes 1 after the referece position return is executed.

The operation of Cs contour control axis (Manual/Automatic)

If the Cs contour control axis has been returned to the reference position, the operation of Cs axis is same as the common NC axis.

In the spindle speed control, the Cs contour control axis can not be performed. Otherwise, the system alarms.

So, in the spindle speed control mode, it is not permitted the manual operation of Cs by the PLC ladder diagram.

The signal shift of spindle contour control

CON (G027#7)

[Type] Signal input

[Function] This signal is used for shifting between spindle speed control mode

and Cs contour control mode.

When this signal is set to 1, the spindle is shifted to the Cs contour control mode from speed control mode.

When this signal is set to 0, the Cs contour control mode comes back to the speed control mode.

The signal shift end of spindle contour control

FSCSL(F044#1)

[Type] Signal output

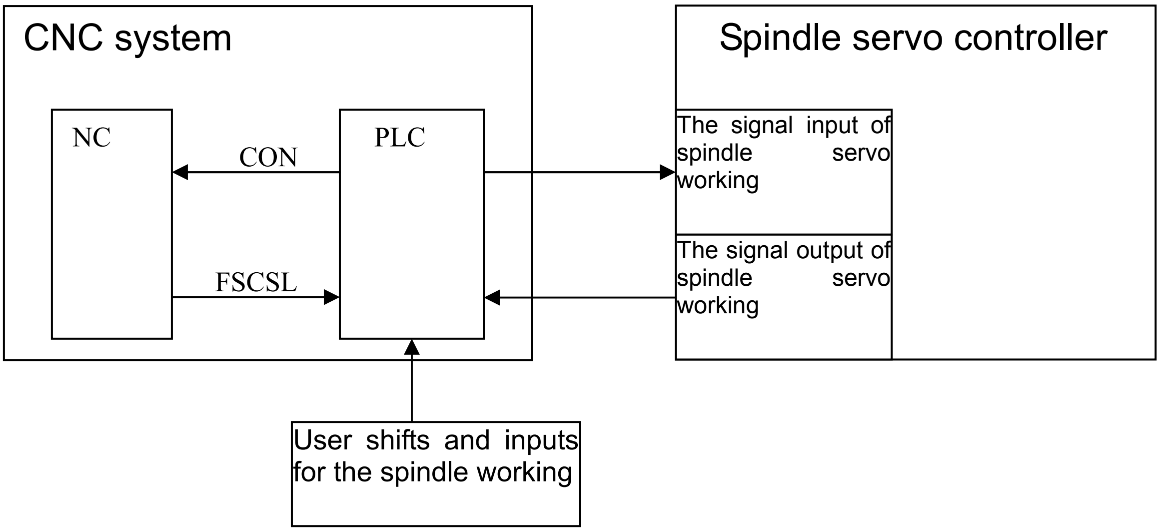
[Function] This signal indicates that the controlled axis has been controlled under the Cs contour.

[Output condition] Spindle speed control mode —> 0

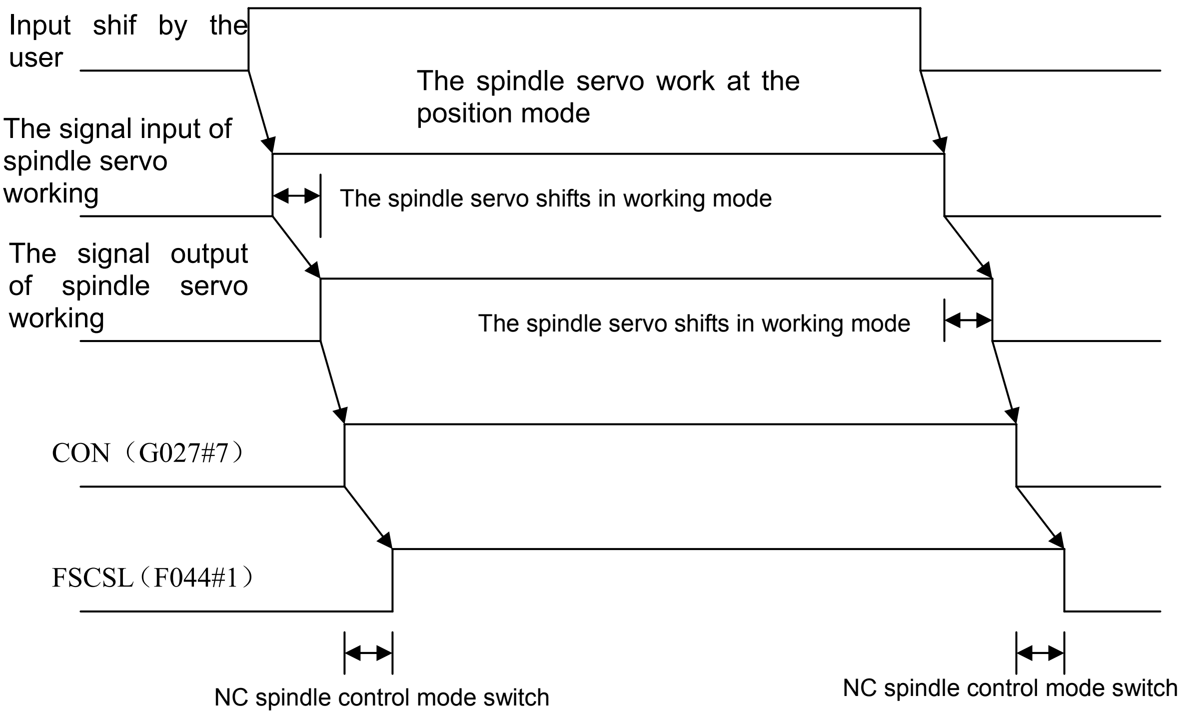
Cs contour control mode —> 1

CNC and spindle servo control unit

The signal shift relationship of the spindle working



Time sequence figure



Relative parameter

0	7	7	The start speed of acceleration/deceleration of CS axis

Resolution range: 0～5000 (Unit:deg/min)

0	7	8	The acceleration/deceleration time constant of CS axis

Resolution range: 10～4000 (Unit: ms)

- **The explanation of "two points same"**

Radius compensation mode is pre-read two blocks. Caculate the transit point and perform a path movement taking 3 position points (the start of the 1st block, the intersection of the 1st and the 2nd blocks, the end of the 3rd block). In this case, "two same points" may occur in the following items:

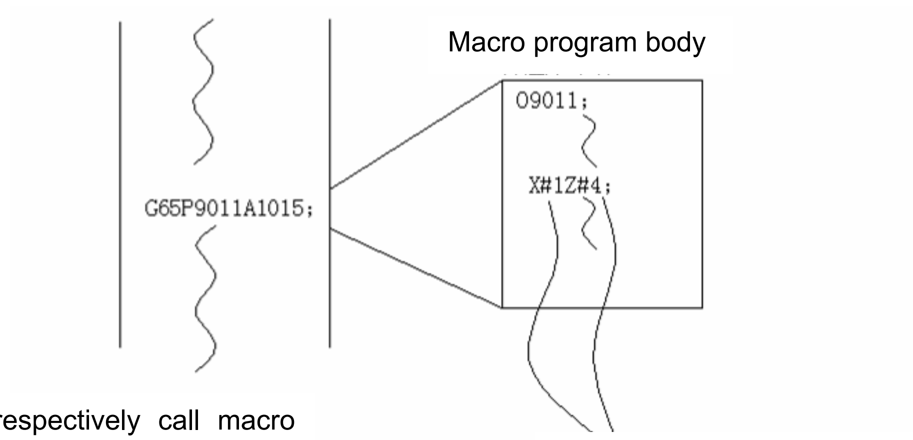
(a) The first two points are same when starting.
(b) The last two points are same when starting.
(c) The first two points are same during the compensation.
(d) The last two points are same during the compensation.
(e) The first two points are same during the retraction.
(f) The last two points are same during the retraction.

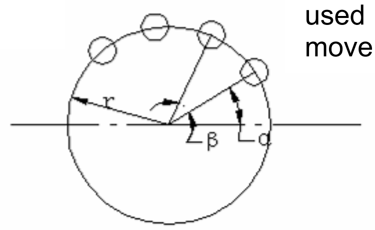
The "two same points" is regarded the point as a linear of which approximates to zero, when the "two same points" occurs, the transit point calculation can be performed based on the straight line (point) to straight line (point), straight line (point) to circular arc (point), circular arc (point) to straight line (point) and circular arc (point) to circular arc (point).

CHAPTER 5 MACRO PROGRAM

GSK980MDa provides macro programs which is similar to high level language. Variable assignment, arithmetic operation, logical judgment and conditional branch can be realized through custom macro program. It is in favor of the programming for special parts, lessens the complex operation and simplifies the custom program.

Custom macro programs are similar to subprograms. However, macro program allows variable assignment, arithmetic operation, logical judgment and conditional branch, which makes it easier to program the same machining process.





It is easy to machine the screw holes distributed in circles (shown in the figure above).

After a macro program used in circular holes is programmed and edited, it can be performed if the NC system has circular hole machining function.

By the following command, programming personnel can use circular holes function.

G65 P p R r A a B b K k ;

p: Macro program number of circular holes

r: Radius

a: Start angle of the hole

b: Angle of holes intervals

k: Holes number

In this way, users can improve the NC performance on their own. Macro programs can be either provided by machine tool builder or defined by users.

5.1 Macro Call

Macro call (G65, G66) differs from subprogram call (M98) as described below:

1. With G65 or G66, an argument (data passed to a macro) can be specified. M98 does not have this capability.

2. When an M98 block contains another NC command (for example, G01 X100.0 M98 P), the macro program P_ is called after the command G01 is executed. On the other hand G65 unconditionally calls a macro P_.

3. When an M98 block contains another NC command (for example,G01 X100.0 M98 P_), the machine stops in the single block mode. On the other hand, G65 does not stop the machine.

4. With G65 or G66, the level of local variables changes. With M98, the level of local variables does not change.

- **Non-modal call（G65）**

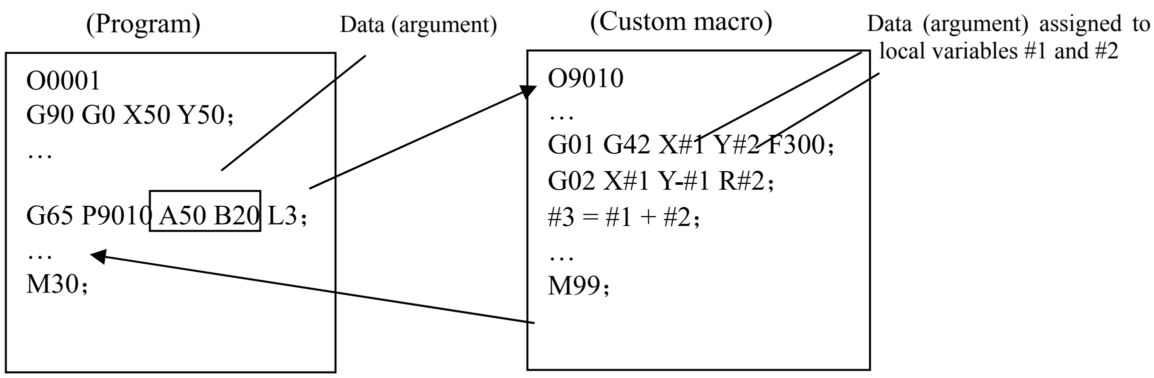
When G65 is specified, the macro program specified at address P is called. Argument (data) can be passed to the custom macro program.

Format： G65 P_ L_ <argument>_；

Explanation： P —— number of the program to be called

L —— repetition count（1 by default, 1 to 9999 can be specified）

<Argument> —— Data passed to the macro. Its value is assigned to the corresponding local variables.



Argument specification: two types of argument specification are available.

Argument specification Ⅰ：it uses letter other than G, L, O, N and P once each. In repeated specification, the last one prevails.

Argument specification I

Address	Variable number
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
H	#11

Address	Variable number
I	#4
J	#5
K	#6
M	#13
Q	#17
R	#18
S	#19

Address	Variable number
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

Note: Addresses that need not to be specified can be omitted. Local variables corresponding to an omitted address are set to null.

Argument specification II: Uses A, B, C and Ii, Ji, Ki (i is 1~10) and automatically decides the argument specification type according to the letters and the sequence. Uses A, B, C once each and uses I, J, and K up to ten times.

Argument specification II

Address	Variable number	Address	Variable number	Address	Variable number
A	#1	K_3	#12	J_7	#23
B	#2	I_4	#13	K_7	#24
C	#3	J_4	#14	I_8	#25
I_1	#4	K_4	#15	J_8	#26
J_1	#5	I_5	#16	K_8	#27
K_1	#6	J_5	#17	I_9	#28
I_2	#7	K_5	#18	J_9	#29
J_2	#8	I_6	#19	K_9	#30
K_2	#9	J_6	#20	I_{10}	#31
I_3	#10	K_6	#21	J_{10}	#32
J_3	#11	I_7	#22	K_{10}	#33

Note 1: Subscripts of I, J and K for indicating the order of argument specification are not written in the actual program.

Note 2: Argument I, J, K do not need to be written in orders. They will be identified according to the present sequence. For example: G65 P9010 A1 B2 C3 I14 J15 I6 J7 K9 K11 K12 J30; The variables are passed as follows:

I14→#4, J15→#5, I6→#7, J7→#8, K9→#6, K11→#9, K12→#12, J30→#11;

Format: G65 must be specified before any argument.

Mixture of argument specifications I and II: The CNC internally identifies argument specification I and II. If a mixture of argument specification I and II is specified, the type of argument specification specified later take precedence.

Example

G65 P9001 A1.2 B2.0 I-3.3 I4 D5;

<variable>

#1: 1.2

#2: 2.0

#3: Null

#4: -3.3

#5: Null

#6: Null

#7: 4

5

When both I4 and D5 arguments are commanded for variable # 7 in this example, the later, D5 is valid.

- **Modal call (G66)**

Once G66 is issued to block specifying movement along axes is executed. This continues until G67 is issued to cancel a modal call.

Note: The format, functions and argument specification of G65 are identical with that of the G65 (non-modal call). (Refer to the introduction of G65 for detailed description).

Modal call nesting: Modal calls can be nested by specifying another G66 code during

a modal call.

Explanation: 1. In the specified G66 block, only argument is passed, and macro modal call will not be executed.

2. Macro modal call can only be executed in the blocks with G00, G01, G02, and G03

3. No macro program can be called in a block which contains a code such as miscellaneous function that does not involve movement along an axis.

4. G65 and G66 should not be specified at the same time.

5. Multiple macro programs cannot be called in G66 block.

6. As with G65, G66 should be specified prior to arguments and P.

- **Sample program**

  ➢ **G65 call (bolt hole circle)**

Create a macro program for machining holes on a circle. The radius is I; start angle is A; holes interval is B, holes number is H; the center of the circle is (X,Y). Commands can be specified in either the absolute or incremental mode. To drill in the clockwise direction, specify a negative value for B.

Format: G65 P9100 Xx Yy Zz Rr Ii Aa Bb Hh;

X: X coordinate of center point (absolute or incremental) (#24)

Y: Y coordinate of center point (absolute or incremental) (#25)

Z: Hole depth (#26)

R: Coordinates of an rapid approaching point (#18)

F: Cutting feedrate (#9)

I: Circle radius (#4)

A: Drilling start angle (#1)

B: Incremental angle (clockwise when negative value is specified). (#2)

H: Number of holes (#11)

Macro call : O0002

```
G90 G00 X0 Y0 Z100;
G65 P9100 X100 Y50 R30 Z-50 F500 I100 A45 B30 H5;
M30;
```

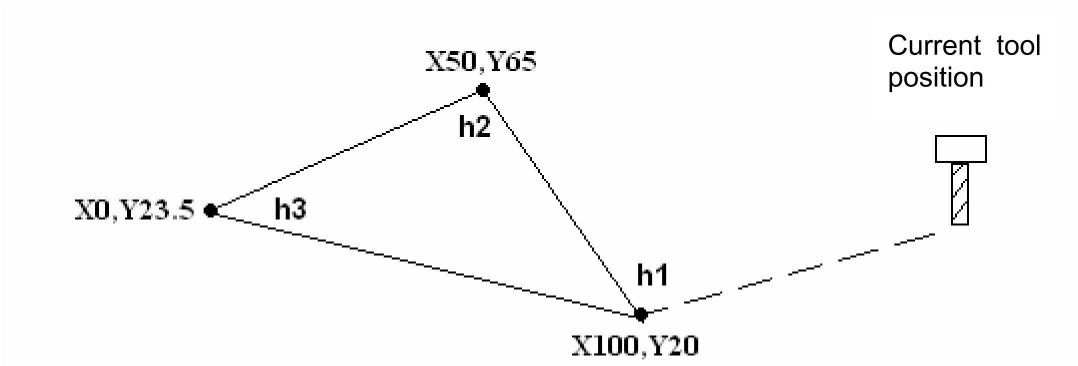
Macro program (the called program): O9100

```
#3=#4003 … ………………………….. Stores G codes of 03 group
IF[#3 EQ 90]GOTO 1; … ………………Branches to N1 in the G90 mode
#24=#5001+#24; … ………Calculates the X coordinate of the center point
#25=#5002+#25; … ………… Calculates the Y coordinate of the center point
N1 WHILE [#11 GT 0] DO 1; … ……Until the number of remaining holes reaches 0
#5=#24+#4*COS[#1]; … ………………Calculates the hole position on X axis
#6=#25+#4*SIN[#1]; … ………………..Calculates the hole position on X axis
G90 G81 X#5 Y#6 Z#26 R#18 F#9; … …Drilling after moving to the target position
#1=#1+#2; … ……………………………Updates the angles
#11=#11-1; … ………………………….…Decrements the number of holes
END 1;
```

G#3 G80; … ……………………Returns the G codes to the original state.
M99;

Argument meanings：#3 store G codes of 03 group
#5 X coordinate of the next hole to drill
#6 Y coordinate of the next hole to drill

➢ **G66 modal call**
Shown as follows: machine 3 holes (h1,h2,h3)



Call format：G66 P9201 Aa Bb Cc; （the argument in this example is assumed）

Macro program： O0001
G90 G17 G00 X0 Y0 Z0；
G00 X150 Y20； ----------------------position
G66 P9201 A-10 B-40 C2000；-----pass the argument, be ready for machining
G00 X100 Y20；-----------------------position to h1, call macro program (hole machining)
G00 X50 Y65；-------------------------position to h1, call macro program (hole machining)
M09； ---------------------non-movement code, does not call macro program
G00 X0 Y23.5；---------position to h1, call macro program (hole machining)
G67；------------------------------------cancel macro program modal call
G00 X150 Y20；------------------------positioning return
M30;

Called macro program：O9201（machining process）
G81 G98 R#1 Z#2 F#3；
M99；

5.2 Variables

An ordinary machining program specifies a G code and the travel distance directly with a numeric value, for example, G01 and X100.0. With a custom macro program, numerical value can be specified directly or using variables, for example, G#101 X#102. When variables are used, the variable value can be changed by programs or using operation on the MDI panel.

- **Representation and using methods of variables**

Differ from argument (data), variables are considered as the carrier of data, for example, #1, #101 …are variables; A100, B200 …are arguments. Data of arguments A100, B200 should be transferred to variable #1 and #2. When using or programming macro programs, numerical value can be specified directly (such as G01, X100) or using variables (such as G#01, X#07). When variables are used, the variable value can be changed by programs or using operation on the panel.

The address value of a macro body can be specified by variables. The variable value can be set by the main program or be assigned the calculated value when executing the macro body. Multiple variables can be identified by numbers.

(1) Variable representation

A number sign # followed by a variable number is shown as follows:

#i (i = 1, 2, 3, 4 ……). For example: #5, #109, #1005

(2). Omission of decimal point

When a variable value is defined in a program, the decimal point can be omitted. For example: when defining #1=123, the actual value of variable #1 is 123.000.

(3). Referencing variables

To reference the value of a variable in a program, specify a word address followed by the variable number. A program with an expression <address>#i or <address>-#i indicates that the variable value or negative value is used as address value.

For example: Z-#110…when #110 = 250, it is equals to Z-250.

G#130…when #130 = 3, it is equals to G3

(4). Replace variable numbers with variables

When replace variable numbers with variables, #9100 rather than ##100 is used, the 9 followed # means the replacement. For example: when #100 = 105, #105 = 500,

X#9100 and X500 are equal. i.e. X#9100 → X##100，X#105 → X500

X-#9100 and X-500 are equal.

Note: Program number o, sequence number N and optional block skip number '/' cannot be followed with variables. For example: O#1, /#2, N#3 .

- **Variable display**

Macro variables 02000 N00000

No.	Data	No.	Data	No.	Data
100	Null	108	108.000	116	Null
101	12.235	109	Null	117	Null
102	110100101	110	Null	118	Null
103	0.000	111	Null	119	Null
104	0.000	112	Null	120	Null
105	Null	113	**********	121	Null
106	Null	114	Null	122	Null
107	Null	115	Null	123	Null

No. 108
EDIT S0000 T01 H00

1. On macro variable page, "Null" indicates the variable is null, i,e, undefined. The mark ********** indicates the variable value overflows of the range (but the internal stored data may not overflow).
2. The value of common variables (#100~#199，#500~#999) can be displayed on macro variable page, or be assigned directly by inputting data on the page.
3. The value of local variables (#1~#33) and system variables do not have display screen. A value of local variable or system variable can be displayed by assigning the value to common variables.
4. Variable data range: integral type: -2147483648~2147483647， real number type：-10^{47}~-10^{-29}, 0, or 10^{-29}~10^{47}.

Intergra type: 2147483648~2147483647 real number type: -10^{47}~-10^{-29}, 0, or 10^{-29}~10^{47}.

- **Types of variables**

Variables are classified into four types by variable number:

Variable number	Type of variable	Function	Range	Remark
#0	Null variable	This variable is always null. No value can be assigned to this variable.	NULL	
#1~#33	Local variable	Local variable can only be used within a macro to hold data such as the results of operations. When the power is turned off, local variables are initialized to null. When a macro is called, arguments are assigned to local variables.		
#100~#199	Common variable	Common variables can be shared among different macro programs.	When the power is turned off, variables	read/ write/

			are initialized to null.	display
#500~#999			When the power is turned off, data is stored	
#1000~#1015	System variable （234）	G54, G55 output	0,1 processed by PLC	Read only
#1032		Store G54, G55, read all 16 bits of a signal at one time		
#1100~#1115		G54, G55 input		Read/write
#1132		Store G54, G55,write all 16 bits of a signal at one time		
#1133		Store G56~G59, write all 32 bits of a signal at one time		
#2001~#2032	System variable	Tool length compensation wear	-9999.999~9999.999	Read/write
#2201~#2232		Tool length compensation	-9999.999~9999.999	Read/write
#2401~#2432		Cutter compensation wear	-9999.999~9999.999	Read/write
#2601~#2632		Cutter compensation wear	-9999.999~9999.999	Read/write
#3003~#3004		Automatic operation control—#3003	0，1，2，3	Read/write
		Automatic operation control—#3004	0~7	Read/write
#3901		The number of machined parts	0~99999999	Read/write
#4001		G00, G01, G02, G03, G73, G74, G80, G81, G82, G83, G84, G85, G86, G88, G89, G110, G111, G112, G113, G114, G115, G134, G135, G136, G137, G138, G139	modal G code group1	Read only
#4002~#4003		G17, G18, G19—#4002	modal G code group 2	Read only
		G90, G91—#4003	modal G code group 3	Read only
#4005~#4007		G94, G95—#4005	modal G code group 5	Read only
		G20, G21—#4006	modal G code group 6	Read only
		G40, G41, G42—#4007	modal G code group 7	Read only
#4008		G43, G44, G49	modal G code group 8	Read only
#4010		G98, G99	modal G code group 10	Read only
#4014		G54~G59	modal G code group	Read

			14	only
#4107		D code	0~32	Read only
#4109		F code	0~15000	Read only
#4111		H code	0~32	Read only
#4113~#4115		M code—#4113	0~99	Read only
		Sequence number—#4114	0~99999	Read only
		Program number —#4115	0~9999	Read only
#4119~#4120		S code—#4119	0~9999	Read only
		T code—#4120	0~32	Read only
#5001~5005	System variable	1~5 axes; block end point; workpiece coordinate system; tool compensation value not included	-9999.999~9999.999	Read only
#5021~5025		1~5 axes; current position; machine coordinate system; tool compensation value included	-9999.999~9999.999	Read only
#5041~5045		1~5 axes, the current position, workpiece coordinate system contain tool compensation value	-9999.999~9999.999	Read only
#5061~5065		1~5 axes, skip signal position; workpiece coordinate system; tool compensation value included	-9999.999~9999.999	Read only
#5081~5085		1~5 axes; tool length compensation value; current execution value.	-9999.999~9999.999	Read only
#5201~5205		1~5 axes; external workpiece zero point offset value	-9999.999~9999.999	Read/write
#5221~5225		1~5 axes, G54 workpiece zero point offset value	-9999.999~9999.999	Read/write
#5241~5245		1~5 axes, G55 workpiece zero point offset value	-9999.999~9999.999	Read/write
#5261~5265		1~5 axes, G56 workpiece zero point offset value	-9999.999~9999.999	Read/write
#5281~5285		1~5 axes, G57 workpiece zero point offset value	-9999.999~9999.999	Read/write
#5301~5305		1~5 axes, G58 workpiece zero point offset value	-9999.999~9999.999	Read/write
#5321~5325		1~5 axes, G59 workpiece zero point offset value	-9999.999~9999.999	Read/write

5.2.1 Null Variables

When the variable value is undefined, the variable is null. Variable #0 is always null, and can be read only.

a, referencing

The address itself is ignored when an undefined variable (null variable) is quotated.

When #1=< Null>,	**When #1=0**
G90 X100 Y#1 equals to G90 X100	G90 X100 Y#1 equals to G90 X100 Y0

b, Arithmetic operation

<Null> equals to 0 in any case except when assigned by <Null>.

When #1=< Null >	**When #1=0**
#2=#1 （assignment） The arithmetic operation result #2 equals to< Null>	#2=#1 The arithmetic operation result #2 equals to 0
#2=#1 * 5 The arithmetic operation result #2 equals to 0	#2=#1 * 5 The arithmetic operation result #2 equals to 0
#2=#1+#1 The arithmetic operation result #2 equals to 0	#2=#1+#1 The arithmetic operation result #2 equals to 0

c. Conditional expression

<Null> differs from 0 only for EQ and NE.

When #1= Null	**When #1=0**
#1 EQ #0 ↓ True	#1 EQ #0 ↓ False
#1 NE #0 ↓ False	#1 NE #0 ↓ False
#1 GE #0 ↓ False	#1 GE #0 ↓ False
#1 GT #0 ↓ False	#1 GT #0 ↓ False

5.2.2 Local Variables

Local variables are the variables internally defined in a program. They are effective only within the program, i.e., it is only can be used within the program.

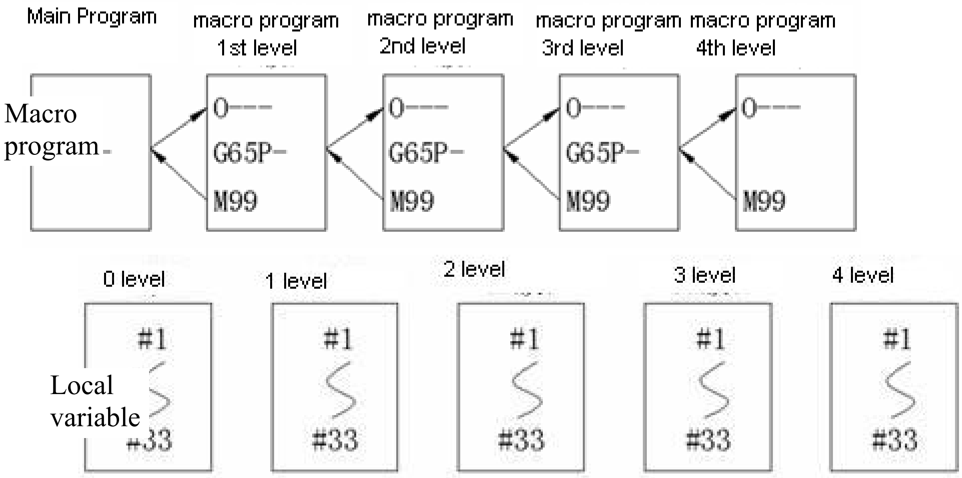
A local variable #1 that calls macro programs at a certain moment is different from the #1 at another moment. (No matter the macro programs are identical or not). Therefore, when macro program B is called from macro program A, like nesting, the local variables used in macro A will not be misused in macro B, and will not disable the value in macro B.

Usually, the local variables are used to accept the value passed from argument. Please refer to" Argument Specification" for the relationship between arguments and addresses. Pay attention that, the initial state of local variable is Null, before the local variable is defined (assigned).

- **Custom macro program nesting and local variable**

When calling a macro program, its nesting level increases by one, and correspondingly, the level of local variable increases by one as well.

The relationship between macro program call and local variable is shown as follows:



- **Explanations**

1. #1～#33 local variables (0 level) are provided in the main program.

2. When a macro program (1 level) is called by G65, the local variable (0 level) is stored, and local variables #1~#33 of the new macro program is prepared. The argument replacement is possible (the same as ③).

3. Each time a macro program (2, 3, 4 levels) are called, local variables (1, 2,3 levels) in each group are stored, and new local variables (2,3,4, levels) are prepared.

4. When M99 (return from macro programs) is commanded, the local variables (0, 1, 2, 3 levels) stored in ②, ③ are recovered in the state as they are stored.

5.2.3 Common Variable

Common variable is the global variable defined within the system. It can be used in any program. That is to say, #101 used in a macro program is the same as the one used in another macro program. Therefore, the arithmetic operation result of common variable #101 in a program can be used in another program.

In the system, there is no special regulation for using common variables. #100~#199 is the variable group without power-off memory function; #500~#999 is the variable group with power-off memory function, i.e. data are stored after power-off.

5.2.4 System Variables

System variables are used to read and write CNC internal data, such as tool length compensation value, tool nose radius compensation value. Some system variables can only be read. System variables are the basis of automatic control and general-purpose machining program development.

- **Interface signal** The macro variable corresponding to interface signal is the exchange signal between PLC and custom macro program.

Variable No.	Function
#1000~#1015	A 16-bit signal can be sent from the PLC to a custom macro. Used to read signal bit by bit.
#1032	A 16-bit signal can be sent from the PLC to a custom macro. Used to read al 16 bits of a signal at one time.
#1100~#1115	A 16-bit signal can be sent from the PLC to a custom macro. Used to read and write signal bit by bit.
#1132	A 16-bit signal can be sent from the PLC to a custom macro. Used to read and write all 16 bits of a signal at one time.
#1133	A 32-bit signal can be sent from the PLC to a custom macro. Used to read all 32 bits of a signal at one time.

Note: Please refer to the *GSK980TD PLC User Manual* for the relationships between variables and F, G signals.

- **Tool compensation value** tool compensation value can be read and written

Compensation No.	Tool length compensation		Cutter compensation	
	Geometric（H）	Wear （H）	Geometric（D）	Wear （D）
01	#2201	#2001	#2601	#2401
02	#2202	#2002	#2602	#2402
03	#2203	#2003	#2603	#2403
…….				
31	#2231	#2031	#2631	#2431
32	#2232	#2032	#2632	#2432

● **Automatic operation control** The control state of automatic operation can be changed

Variable No.	Variable value	Single block	Completion of an auxiliary function
#3003	0	Enabled	To be awaited
	1	Disabled	To be awaited
	2	Enabled	Not to be awaited
	3	Disabled	Not to be awaited

Note 1: When the power is turned on, the value of this variable is 0.

Note 2: When single block stop is enabled (G46.1 is 1), the state of #3003 can change the execution of single block stop.

Note 3: When single block stop is disabled (G46.1 is 0), single block stop operation is not performed even if the single block switch is set to ON.

Note 4: When a wait for the completion of auxiliary function (M, S and T functions) is not specified, program execution proceeds to the next block before completion of auxiliary functions. Also distribution completion signal DEN is not output.

Variable No.	Variable value	Feed hold	Feedrate override	Exact stop
#3004	0	Enabled	Enabled	Enabled
	1	Disabled	Enabled	Enabled
	2	Enabled	Disabled	Enabled
	3	Disabled	Disabled	Enabled
	4	Enabled	Enabled	Disabled
	5	Disabled	Enabled	Disabled
	6	Enabled	Disabled	Disabled
	7	Disabled	Disabled	Disabled

Note 1: When the power is turned on, the value of this variable is 0.

Note 2: When feed hold is disabled, if the feed hold button is held down, the machine stops in the single block stop mode. However, single block stop operation is not performed when the single block mode is disabled with variable #3003.

Note 3: When the feed hold is disabled, if the feed hold button is pressed then released, the machine does not stop; program execution continues and the machine stops at the first block where feed hold is enabled; the feed hold lamp is ON.

Note 4: When feedrate override is disabled, an override of 100% is always applied regardless of the setting of the feedrate override.

Note 5: When exact stop check is disabled, no exact stop check is

made even in blocks including those which do not perform cutting.

- **Number of machined parts** The number of machined parts can be read and written.

Variable No.	Function
#3901	Number of machined parts

- **Modal information**

  Modal information specified in blocks up to the immediately preceding block can be read.

Variable No.	Function
#4001	Group 1 （G00, G01, G02, G03, G73, G74, G80, G81, G82, G83, G84, G85, G86, G88, G89, G110, G111, G112, G113, G114, G115, G134, G135, G136, G137, G138, G139）
#4002	Group 2（G17, G18, G19）
#4003	Group 3（G90, G91）
#4005	Group 5（G94, G95）
#4006	Group 6（G20, G21）
#4007	Group 7（G40, G41, G42）
#4008	Group 8（G43, G44, G49）
#4010	Group 10（G98, G99）
#4014	Group 14（G54, G55, G56, G57, G58, G59）
#4107	D code
#4109	F code
#4111	H code
#4113	M code
#4114	Block sequence number
#4115	Program name
#4119	S code
#4120	T code

- **Current position** Position information can be read.

Variable No.	Function	Read during movement
#5001~#5005	Workpiece coordinate system block end point (tool compensation value not included)	Enabled
#5021~#5025	Machine coordinate system current position(tool compensation value	Disabled

	included)	
#5041~#5045	Workpiece coordinate system current position (tool compensation value included)	Disabled
#5061~#5065	Workpiece coordinate system skip signal position (tool compensation value included)	Enabled
#5081~#5085	Tool length compensation value	Disabled

Note 1: The first digit (from 1 to 5) represents an axis number.

Note 2: The tool length compensation value currently used for execution rather than the immediately preceding tool compensation value is held in variables #5081~#5085.

- **Workpiece coordinate system compensation value**

Workpiece coordinate system compensation value can be read and written.

Variable No.	Function
#5201~#5205	The first to the fifth axes external workpiece zero point offset value
#5221~#5225	The first to the fifth axes G54 workpiece zero point offset value
#5241~#5245	The first to the fifth axes G55 workpiece zero point offset value
#5261~#5265	The first to the fifth axes G56 workpiece zero point offset value
#5281~#5285	The first to the fifth axes G57 workpiece zero point offset value
#5301~#5305	The first to the fifth axes G58 workpiece zero point offset value
#5321~#5325	The first to the fifth axes G59 workpiece zero point offset value

5.3 Arithmetic and Logic Operation

- <u>Macro programs in both traditional G65 H format and statement format are compatible with GSK980MDa.</u>

  <u>Users can alternatively select one of them for programming. This makes programming more convenient and flexible.</u>

- <u>Please strictly observe the formats and specifications in the following “Arithmetic and Logic Operation” table.</u>

Arithmetic and Logic Operation

Function	**Statement format**	**Traditional G65H format**	**Remark**
Definition, assignment	#i = #j	G65 H1 P#i Q#j	
Sum Subtraction	#i = #j + #k #i = #j - #k	G65 H2 P#i Q#j R#k G65 H3 P#i Q#j R#k	Logic operation is performed on binary

Multiplication Division	#i = #j * #k #i = #j / #k	G65 H4 P#i Q#j R#k G65 H5 P#i Q#j R#k	numbers bit by bit.
OR AND XOR	#i = #j OR #k #i = #j AND #k #i = #j XOR #k	G65 H11 P#i Q#j R#k G65 H12 P#i Q#j R#k G65 H13 P#i Q#j R#k	
Square root Absolute value Rounding off Rounding up Rounding down Nature logarithm Exponential function	#I = SQRT ［#j］ #I = ABS ［#j］ #I = ROUND ［#j］ #I = FUP ［#j］ #I = FIX ［#j］ #I = LN ［#j］ #I = EXP ［#j］	G65 H21 P#i Q#j G65 H22 P#i Q#j G65 H23 P#i Q#j G65 H24P#i Q#j G65 H25 P#i Q#j G65 H26 P#i Q#j G65 H27 P#i Q#j	
Sine Arcsine Cosine Arccosine Tangent Arctangent	#i = SIN ［#j］ #i = ASIN ［#j］/［#k］ #i = COS ［#j］ #i = ACOS ［#j］ #i =TAN ［#j］ #i = ATAN［#j］/［#k］	G65 H31 P#i Q#j G65 H32 P#i Q#j G65 H33 P#i Q#j G65 H34 P#i Q#j G65 H35 P#i Q#j G65 H36 P#i Q#j R#k	An angle is specified in degree. 90 degrees and 30 minutes is represented as 90.5 degree.
Conversion from BCD to BIN Conversion from BIN to BCD	#i = BIN ［#j］ #i = BCD ［#j］	G65 H41 P#i Q#j G65 H42 P#i Q#j	Used for the signal exchange to and from PLC.

Unconditional branch
Equals to branch
Not equals to branch
Greater than branch
Smaller than branch
Greater than or equals to branch
Smaller than or equals to branch
	GOTO #i
	IF (#i EQ #j) GOTO #k
	IF (#i NE #j) GOTO #k
	IF (#i GT #j) GOTO #k
	IF (#i LT #j) GOTO #k
	IF (#i GE #j) GOTO #k
	IF (#i LE #j) GOTO #k
		G65 H80 P#i Q#j R#k
		G65 H81 P#i Q#j R#k
		G65 H82 P#i Q#j R#k
		G65 H83 P#i Q#j R#k
		G65 H84 P#i Q#j R#k
		G65 H85 P#i Q#j R#k
		G65 H86 P#i Q#j R#k
			Please note that #K is the skip signal in macro statement and P#i is the skip signal in traditional G65H format.

User alarm	None	G65 H99 P#i	0≤P≤100

5.3.1 Tranditional Format

If traditional G65 H format is used for programming, only limited operations and jump command can be specified by it. The currently used H operation needs at most 3 operands, so the corresponding operation can be completed when the needed variables (or constants) are obtained in a block.

- **General format**

  G65 Hm P#i Q#j R#k ；

  m: 01～99 means operation command or jump command function

  #i: the name of variable that stored the operation result

  #j: operand 1; it can be constant.

#k: operand 2; it can be constant.

Meaning: #i = #j ○ #k

└──── Operational sign, designated by Hm

(Example) G65 Hm P#100 Q#101 R#102……#100 = #101 ○ #102 ;

G65 Hm R#100 P#101 Q15 ……#101 = 15 ○ #100 ;

G65 Hm Q#100 R-100 P#102……#102 = #100 ○ -100 ;

Note 1: G65 H should be commanded prior to operation or jump command.

Note 2: when P code is commanded in G65 block, G65 P means macro program call. H means argument. No operation or jump command is performed.

Note 3: At most 4 decimal numbers of the constant decimal part can be obtained for rounding. 3 digit numbers can be displayed in the window.

- **Code function explanation**

(1) Variable value assignment, #I ＝ #J

G65 H01 P#I Q#J；

(example) G65 H01 P#101 Q125； (#101 ＝ 125)

G65 H01 P#101 Q#110； (#101 ＝ #110)

G65 H01 P#101 Q-#102； (#101 ＝ -#102)

(2) Addition operation #I ＝ #J ＋ #K

G65 H02 P#I Q#J R#K；

(example) G65 H02 P#101 Q#102 R15； (#101 ＝ #102 ＋ 15)

G65 H02 P#101 Q#110 R#102； (#101 ＝ #110 ＋ #102)

(3) Subtraction operation #I ＝ #J － #K

G65 H03 P#I Q#J R#K；

(example) G65 H03 P#101 Q#102 R#103； (#101 ＝ #102 － #103)

(4) Multiplication operation #I ＝ #J × #K

G65 H04 P#I Q#J R#K；

(example) G65 H04 P#101 Q#102 R#103； (#101 ＝ #102 × #103)

(5) Division operation #I ＝ #J ÷ #K

G65 H05 P#I Q#J R#K；

(example) G65 H05 P#101 Q#102 R#103； (#101 ＝ #102 ÷ #103)

Note: The divisor #k cannot be 0, otherwise an alarm occurs.

(6) OR operation #I ＝ #J OR #K

G65 H11 P#I Q#J R#K；

(example) G65 H11 P#101 Q#102 R#103； (#101 ＝ #102 OR #103)

(7) AND operation #I ＝ #J AND #K

G65 H12 P#I Q#J R#K；

(example) G65 H12 P#101 Q#102 R#103； (#101 ＝ #102 AND #103)

(8) XOR operation #I ＝ #J XOR #K

G65 H13 P#I Q#J R#K；

(example) G65 H13 P#101 Q#102 R#103； (#101 ＝ #102 XOR #103)

(9) Square root $\#I=\sqrt{\#J}$

G65 H21 P#I Q#J；

(example) G65 H21 P#101 Q#102； (#101 ＝ $\sqrt{\#102}$)

Note: the radicand #J cannot be negative, otherwise, an alarm occurs.

(10) Absolute value #I ＝ ∣#J∣

G65 H22 P#I Q#J；

(example) G65 H22 P#101 Q-102； (#101 ＝ ∣-102∣ #101= 102)

(11) Rounding off #I ＝ ROUND[#J]（ROUND off the first decimal）

G65 H23 P#I Q#J；

(example) G65 H23 P#101 Q1.2359；(#101 ＝ 1.2359 #101=1)

(12) Rounding up #I ＝ FUP[#J]

G65 H24 P#I Q#J；

(13) Rounding down #I ＝ FIX [#J]

G65 H25 P#I Q#J；

With CNC, when the absolute value of the integer produced by an operation on a number is greater than the absolute value of the original number, such an operation is referred to as rounding up to an integer. Conversely, when the absolute value of the integer produced by an operation on a number is less than the absolute value of the original number, such an operation is referred to as rounding down to an integer. Be particular careful when handling negative numbers.

(Example) suppose that #1=1.2,#2= -1.2

When #3=FUP[#1] is executed, 2.0 is assigned to #3

When #3=FIX[#1] is executed, 1.0 is assigned to #3

When #3=FUP[#2] is executed, -2.0 is assigned to #3

When #3=FIX[#2] is executed, -1.0 is assigned to #3

(14) Natural logarithm #I ＝ LN [#J]

G65 H26 P#I Q#J；

(example) G65 H26 P#101 Q#102；（#101 ＝ LN[#102]）

Note: when the antilogarithm #j is zero or smaller, otherwise, an alarm is issued.

(15) Exponential function #I ＝ EXP[#J]

G65 H27 P#I Q#J；

(example) G65 H27 P#101 Q#102；（#101 ＝ EXP [#102]）

(16) Sine #I ＝ SIN[#J] (unit: deg)

G65 H31 P#I Q#J；

(example) G65 H31 P#101 Q#103； (#101＝SIN[#103])

(17) Arcsine #I ＝ ASIN[#J]

G65 H32 P#I Q#J；

(example) G65 H32 P#101 Q#103； (#101＝ASIN[#103])

Note 1: When the NAT bit of parameter No.015 is set to 0, the output range is 270° ~ 90°
When the NAT bit of parameter No.015 is set to 1, the output range is -90° ~ 90°

Note 2: Arcsine operand J cannot exceed the range -1~1, otherwise, an alarm is issued.

(18) Arccosine #I ＝ COS[#J] (unit：deg)

G65 H33 P#I Q#J；

(example) G65 H33 P#101 Q#103； (#101＝COS [#103])

(19) Arccosine #I ＝ ACOS[#J]

G65 H34 P#I Q#J；

(example) G65 H34 P#101 Q#103； (#101＝ACOS [#103])

Note 1: Arccosine operand J cannot exceed the range -1~1, otherwise, an alarm is issued.

(20) Tangent #I ＝ TAN[#J] (deg)

G65 H35 P#I Q#J；

(example) G65 H35 P#101 Q#103； (#101＝TAN [#103])

Note: #J cannot be equal to Kπ+π/2（K=0, ±1, ±2, ±3 …）, otherwise the result is wrong.

(21) Arctangent #I ＝ ATAN [#J] / [#K] (unit：deg)

G65 H36 P#I Q#J R#K；

(example) G65 H36 P#101 Q#103 R3； (#101＝ATAN [#103] /[3])

Note 1: When the NAT bit of parameter No.015 is set to 0, the output range is 0° ~ 360°
When the NAT bit of parameter No.015 is set to 1, the output range is -180° ~ 180°

(22) Conversion from BCD to BIN #I ＝ BIN[#J]

G65 H41 P#I Q#J；

(example) G65 H41 P#101 Q#102； (#101 ＝ BIN[#102])

(23) Conversion from BIN to BCD #I ＝ BCD[#J]

G65 H42 P#I Q#J；

(example) G65 H42 P#101 Q#102； (#101 ＝ BCD[#102])

(24) Unconditional branch

G65 H80 Pn； **Pn: sequence number**

(example) G65 H80 P120； (Go to N120 block)

(25) Equal to conditional branch

G65 H81 Q#I R#J Pn; **Pn: sequence number, can be variable**

(example) G65 H81 Q#101 R#102 P1000;

When #101 equals to #102, branch to N1000 block; or execut in order.

(26) Not equal to conditional branch

G65 H82 Q#I R#J Pn; **Pn: sequence number, can be variable**

(example) G65 H82 #101 #102 C1000;

When #101 does not equal to #102, branch to N1000 block; or execut in order.

(27) Greater than conditional branch

G65 H83 Q#I R#J Pn; **Pn: sequence number, variable**

(example) G65 H83 Q#101 R#102 P1000;

When #101 is greater than #102, branch to N1000 block; when #101≤#102, execut in order.

(28) Smaller than conditional branch

G65 H84 Q#I R#J Pn; **Pn: sequence number, variable**

(example) G65 H84 Q#101 R#102 P1000;

When #101 is smaller than #102, branch to N1000 block, or execut in order.

(29) Greater than or equals to conditional branch

G65 H85 Q#I R#J Pn; **Pn: sequence number, variable**

(example) G65 H85 Q#101 R#102 P1000;

When #101 is greater than or equals to #102, branch to N1000 block, or execut in order.

(30) Smaller than or equals to conditional branch

G65 H86 Q#I R#J Pn; **Pn: sequence number, variable**

(example) G65 H86 Q#101 R#102 P1000;

When #101 is smaller than or equals to #102, branch to N1000 block, or execut in order.

(31) P/S alarm issued

G65 H99 Pn; **Pn: sequence number, variable（alarm No.=n +600）**

(example) G65 H99 P15;

P/S custom alarm 615 is issued.

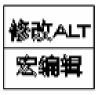
5.3.2 Macro Statement

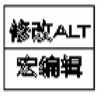
The operations listed in “Arithmetic and Logic Operation” table can be executed in program. The expressions right to the operator contain constants and (or) variables that consisting of functions and operators. The variables #j and #k in the expression can be assigned as constants. The left variable (the first variable) can be assigned by expression. The macro statement is more intuitive, convienent and flexible. It can perform compound operation and multinesting. Sometimes, a macro statement is equal to several tranditional G65H macro programs.

- **General format**

  Please refer the statement format in the “Arithmetic and Logic Operation” table for editing macro statement.

- **Macro program editing**

  In program editing mode or MID mode, by pressing 



key, macro editing state can be switched or inserted.

Differences of two states	**Automatic space**	**Processing of letter O**	**Input of special signs**
Insert state	When editing, spaces are automatically added to identify the words.	Press O to switch, copy, delete programs	Special signs cannot be input
Macro editing state	space are not automatically added	Input as a letter “O”	Special signs can be input

- **Explanations**

1, Angular unit

The angular units of function SIN, COS, ASIN, ACOS, TAN and ATAN are degree. For example, 90°30´ means 90.5 degree.

2, ARCSIN # i=ASIN[#j]

i. the solution ranges are as indicated below

when the NAT bit of parameter No.015 is set to 0: 270°~ 90°

when the NAT bit of parameter No.015 is set to 1: -90°~ 90°

ii. when the #j is beyond the range of -1 to 1, P/S alarm is issued.

iii. a constant can be used instead of the #j variable.

3, ARCCOS # i =ACOS[#j]

i. the solution ranges from 180°~ 0°

ii. when the #j is beyond the range of -1 to 1, P/S alarm is issued.

iii. a constant can be used instead of the #j variable.

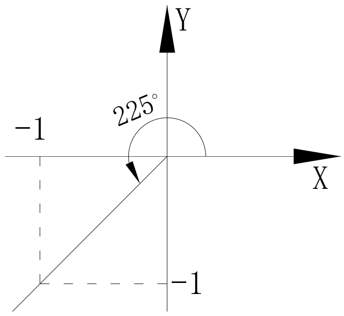
4, ARCTAN #i=ATAN[#j]/[#k]

Specify the lengths of two sides and separate them by a slash /.

The solution ranges are as follows:

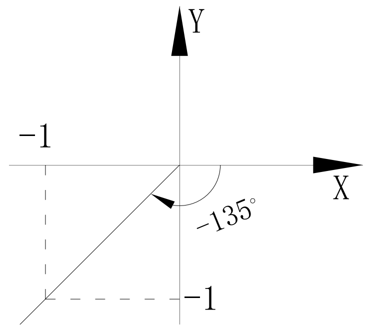
When the NAT bit of parameter No.015 is set to 0: 0°~ 360°

[Example] when #1=ATAN[-1]/[-1] is specified, #1=225°



When the NAT bit of parameter No.015 is set to 1: -180°~ 180°

[Example] when #1=ATAN[-1]/[-1] is specified, #1=-135°



ii. A constant can be used instead of the # j variable.

5. Natural logarithm #i=LN[#j]

i. Note that the relative error may be greater than $10^{-8.}$

ii. When the antilogarithm #j is zero or smaller, P/S alarm is issued.

iii . A constant can be used instead of the #j variable.

6. Exponential function #i=EXP[#j]

i. Note that the relative error may be greater than 10^{-8} .

ii. When the result of the operation exceeds 3.65×10^{47} (j is about 110), an overflow occurs and P/S alarm is issued.

iii. A constant can be used instead of the # j variable.

7, ROUND function

When the ROUND function is included in an arithmetic or logic operation command, IF statement, or WHILE statement, the ROUND function rounds off at the first decimal place.

Example:

When #1=ROUND[#2] is executed where #2=1.2345 the value of variable #1 is 1.0.

When the ROUND function is used in NC statement address, the ROUND function rounds off the specified value according to the least input increment of the address.

8. Rounding up and down to an integer

With CNC, when the absolute value of the integer produced by an operation on a number is greater than the absolute value of the original number, such an operation is referred to as rounding up to an integer. Conversely, when the absolute value of the integer produced by an operation on a number is less than the absolute value of the original number, such an operation is referred to as rounding down to an integer. Be particular careful when handling negative numbers.

Example:

Suppose that #1=1.2, #2= -1.2

When #3=FUP[#1] is executed, 2.0 is assigned to #3.

When #3=FIX[#1] is executed, 1.0 is assigned to #3.

When #3=FUP[#2] is executed, -2.0 is assigned to #3.

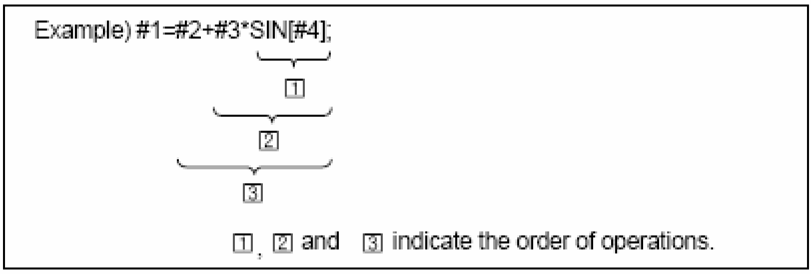
When #3=FIX[#2] is executed, -1.0 is assigned to #3.

5.3.3 Priority of Operations

1. Function

2. Operation such as multiplication and division（*, /, AND）

3. Operation such as addition and subtraction （+, -, OR, XOR）



5.3.4 Bracket Nesting

Brackets are used to change the order of operations. Brackets can be used to multinesting.

Note that the square bracket [,] is used to enclose an expression; the round bracket（，）is used in comments. When the priority is not defined, it is advised to use square bracket to enclose.

5.4 Branch and Repetition

In a program, the flow of control can be changed using the GOTO statement and IF statement. Three types of branch and repetition operations are used:

1. GOTO statement (unconditional branch)
2. IF statement (conditional branch: IF…THEN…)
3. WHILE statement (repetition WHILE…)

5.4.1 Unconditional Branch (GO TO statement)

Go to the block with sequence number n. when a sequence number out the range of 1~99999 is specified, an alarm is raised. A sequence number can also be specified using an expression.

Format: GOTO n； n: sequence number（1~99999）

Example：GOTO 1；GOTO #101；

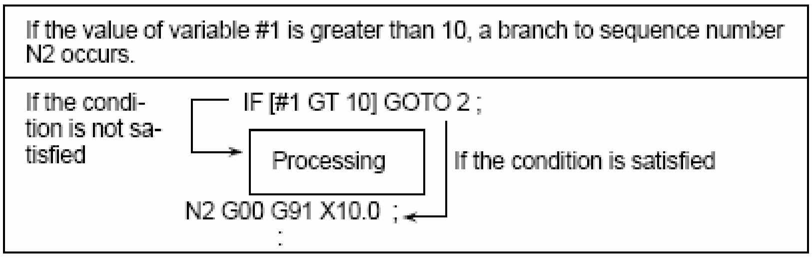
5.4.2 Conditional Branch (IF statement)

Specify a conditional expression after IF.

GOTO format: IF [conditional expression] GOTO n；

If the specified conditional expression is satisfied, a branch to sequence number n occurs. If the specified condition is not satisfied, the next block is executed.

Example:



THEN format：IF [conditional expression] THEN<macro statement>；

If the specified conditional expression is satisfied, a predetermined macro statement is executed. Only a single macro statement is executed.

Example:

IF[#1 EQ #2] THEN #3=0；

If the value of #1 and #2 are the same, 0 is assigned to #3; if not, no execution will be performed.

5.4.3 Conditional Expression

Conditional expression: A conditional expression must include an operator between two variables or between a variable and constant, and must be enclosed in brackets [,]. An expression can be used instead of a variable.

Operators: In 980MDa, operators in the following table are used to compare two values to determine whether they are equal or one value is smaller or greater than the other value.

Operator	Meaning
EQ or = =	Equal to（=）
NE or <>	Not equal to （≠）
GT or >	Greater than（ >）
GE or >=	Greater than or equal to （≥）
LT or <	Less than （<）
LE or <=	Less than or equal to （≤）

Example：IF [3<>2] GOTO 2; it means if 3 is not equal to 2, branch to N2 block

IF [#101>=7.22] THEN #101=SIN30; it means, if #101 is greater than 7.22, the expression after THEN is executed, i.e., assign Sin 30°to #101.

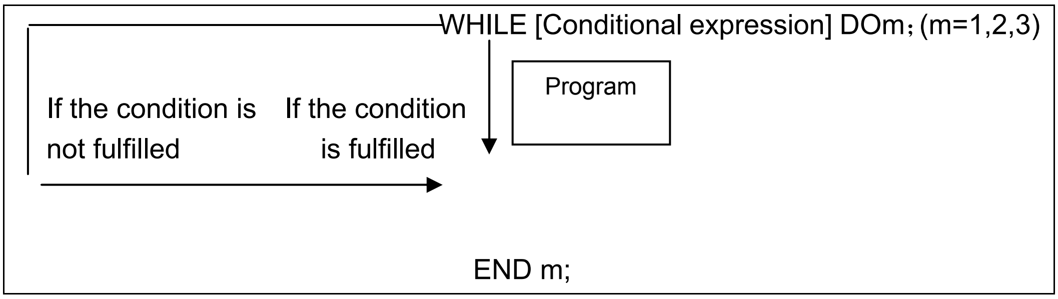
Sample program The sample program below finds the sum of number 1 to 10.

```
O9500
#101=0                  Initial value of the variable to hold the sum
#102=1                  initial value of the variable as an addend
N1 IF[#102 GT 10]GOTO 2 … … Branch to N2 when the
                           addend is greater than 10
#101= #101+#102  … … calculation to find the sum
#102= #102+1  … … Next addend
GOTO 1  … … Branch to N1
N2 M30  … … End of program; Sum of number 1 to 10
```

5.4.4 Repetition（WHILE Statement）

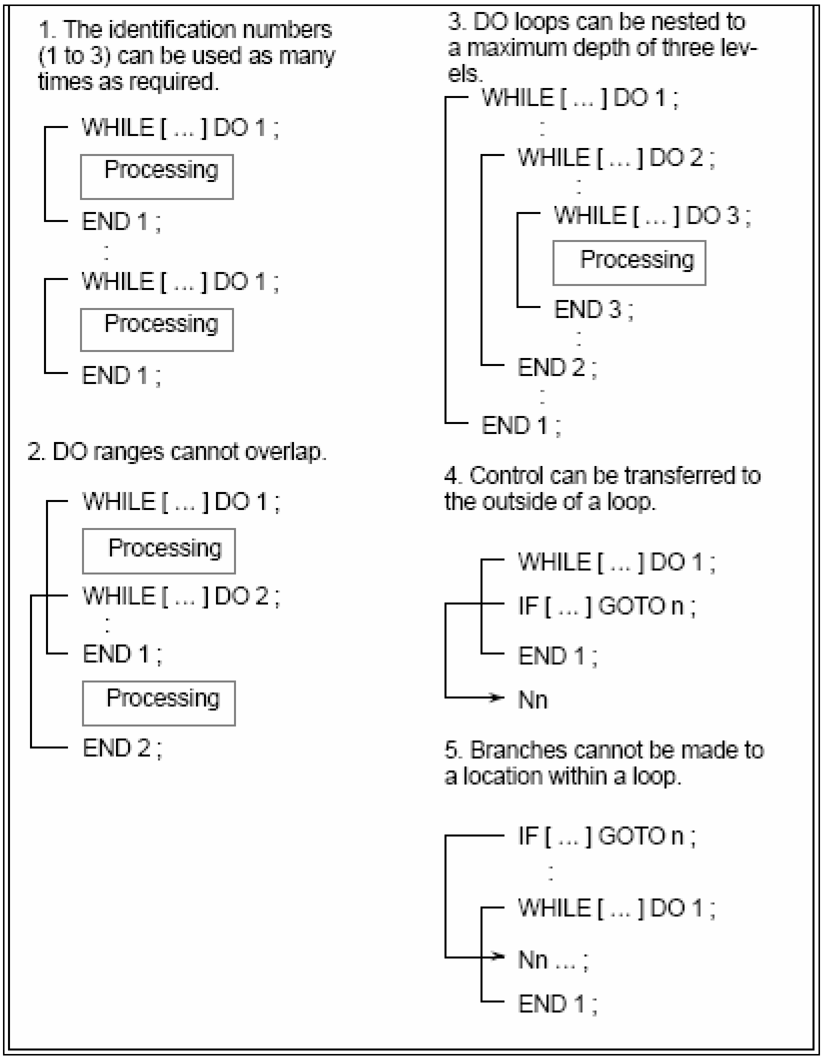
Specify a conditional expression after WHILE. While the specified condition is satisfied, the program from DO to END is executed. If not, program execution proceeds to the block after END.

Example：



Explanations: While the specified condition is fulfilled, the program from DO to END after WHILE is executed. If the specified condition is not fulfilled, program execution proceeds to the block after END. The same format as the IF statement applies. A number after DO and a number after END are identification numbers for specifying the range of execution. The number 1, 2, and 3 can be used. When a number other than 1, 2, and 3 is used, P/S alarm occurs.

Nesting: The identification number (1 to 3) in a DO, END loop can be used as many times as desired. Note, however, when a program includes crossing repetition loops (overlapped DO ranges), P/S alarm occurs.



5.5 Macro Statement and NC statement

The following blocks are referred to as macro statements:

- Blocks containing arithmetic or logic operation (=).
- Blocks containing a controlling statement (such as GOTO, DO, END…)
- Blocks containing a macro call command. (such as G65, G66)

Blocks other than macro statements are referred to as NC statement.

5.5.1 Macro Programming and Registering

Custom macro program are similar to subprogram. They can be edited, registered and used in the same way as subprogram. M98 can call a custom macro program, but cannot pass arguments.

Usually, the macro program is provided by tool builders, but it can also be programmed by customers. It is not necessary for the customers to remermber all related commands in macro programs besides codes that call macro programs.

5.5.2 Limitation

- **Macro statement processing in cutter compensation C mode**

In cutter compensation C mode (G41, G42), in order to calculate the transmission point, NC prereads the next block. The processing way is not the same as general NC statement.

When a macro statement is executed as a single block, it is the block that does not involve movement. And, **in some cases, it cannot correctly execute compensation** (strictly speaking, such block involves 0 distance of movement).

➢ Jump（GOTO,DO,END）

In cutter compensation C mode, when jump command （GOTO, DO, END）is specified, P/S alarm occurs.

➢ When the move command adopts variables

In cutter compensation C, when the move command (such as G01, X#101) adopts variables, P/S alarm occurs. Because cutter compensation C mode is block preread mode, the end point of the next block is essential for calculating the current transmission point position. Specifying X#101 (an unknown data) does not enable a correct calculation of the current transmission point.

- **Single block operation (MDI)**

In MDI mode, macro programs can be specified, but macro program call cannot be executed.

- **Skip "/"**

A "/" appearing in the middle of an <expression> (enclosed in brackets [ ] on the right-hand side of an arithmetic expression) is regarded as a division operator; it is not regarded as the specified for an optional block skip code.

- **Reset**

A reset operation clears any called states of custom macro programs and subprograms, and cursor returns to the first block of the main program.

CHAPTER 6 CUTTER COMPENSATION

6.1 Application for Cutter Radius Compensation

6.1.1 Brief

Generally, the parts machining process is programmed according to parts drawing in one point on a tool. As for the tool used actually, because of the processing or other requirement, the tool is not an ideal point, but an arc only. The position offset exists between actual cutting point and ideal point when the cutting feed is performed. It may cause over cut or undercut, so the part accuracy will be affected. So, the cutter radius compensation can be used to improve the part accuracy in machining.

The path of part figure can be shifted by a cutter radius, which this method is called B type tool compensation; this is a simply method but the movement path of next block can be processed only after a block is performed, so the phenomenon as over cutting will be generated at the intersection point of two blocks.

In order to settle the above issues and eliminate the error, the Tool compensation C should be setup. When a block is read in, the tool compensation C is not performed immediately but the next block is read in again. Corresponding movement path is calculated according to the point of intersection of two blocks (conjunction vector). The tool compensation C performs more accurate compensation in figure because two blocks are read for processing in advance. See the Fig. 6-1

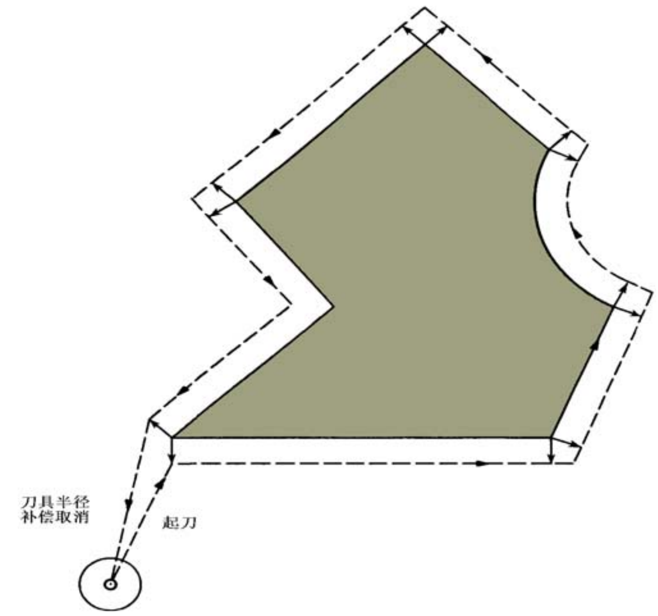


Fig.6-1 C type cutter radius compensation

6.1.2 Compensation value setting

The radius value of each tool should be set before tool compensation C is applied. Tool radius compensation value is set in the OFFSET page (table 6-1), this page contains tool geometric radius and tool radius wear. There into, D is the tool compensation value, when the bit 1 of bit parameter No.003 is 1, the D is compensation value input by diameter. If the bit 1 of bit parameter No.003 is 0, the D is compensation value input by radius. The following explanations are all indicated in radius compensation value if not especially pointed out.

Table 6-1 Display page for CNC cutter radius compensation value

NO.	Geometric（H）	Wearing（H）	Geometric（D）	Wearing（D）
001	20.020	0.030	5.000	0.020
002	10.020	0.123	0.500	0.030
…	…	…	…	…

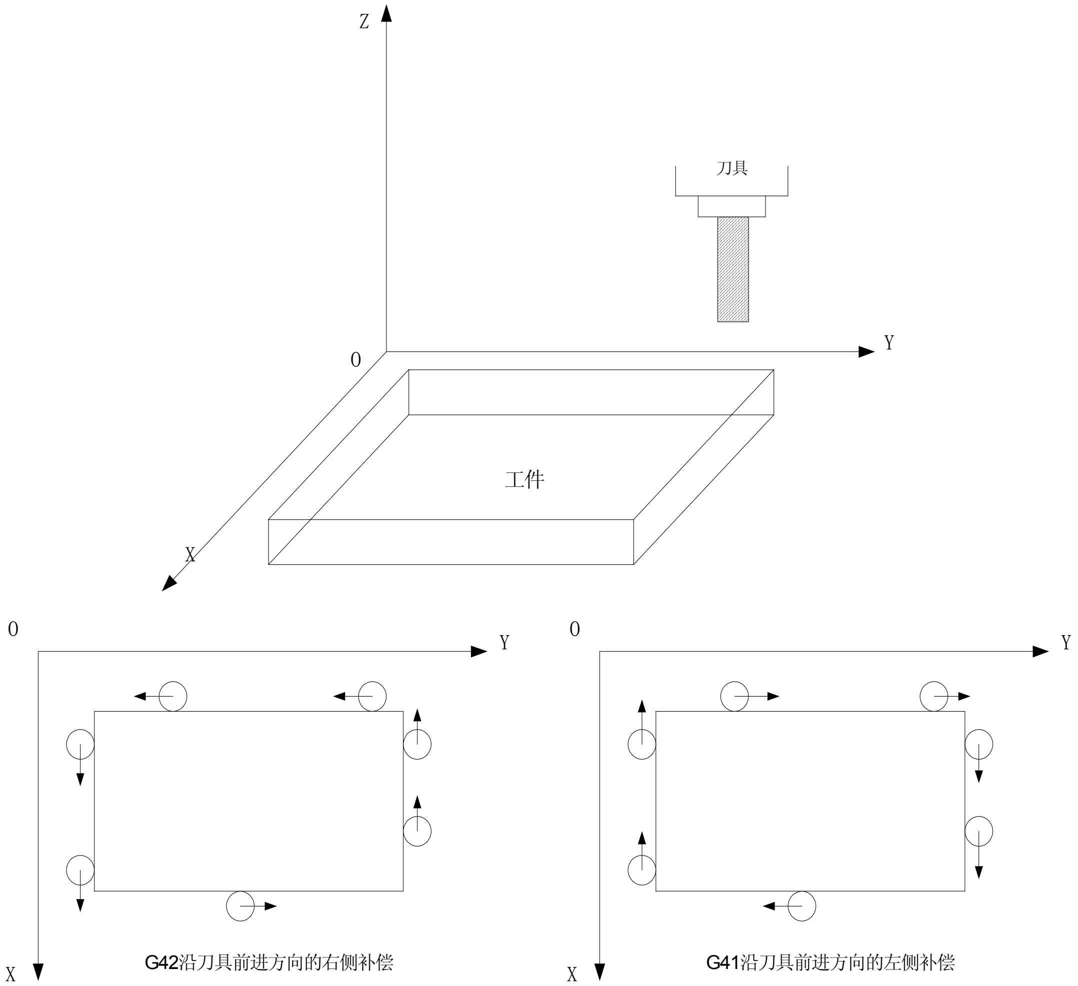
6.1.3 Command format

$$\left\{\begin{matrix}G17\\G18\\G19\end{matrix}\right\}\left\{\begin{matrix}G40\\G41\\G42\end{matrix}\right\}\left\{\begin{matrix}G00\\G01\end{matrix}\right\}\ X\_\ \ Y\_\ \ Z\_\ \ D\_\ ;$$

Commands	Explanation	Remarks
G17	Offset plane selection command (XY plane)	See the Fig.6-2
G18	Offset plane selection command (XZ plane)	
G19	Offset plane selection command (YZ plane)	
G40	Cutter radius compensation cancellation	
G41	Cutter radius compensation left along advancing direction	
G42	Cutter radius compensation right along advancing direction	

6.1.4 Compensation direction

Tool compensation direction is determined according to the relative position of tool with work piece, when the cutter radius compensation is applied. See the Fig.6-2.



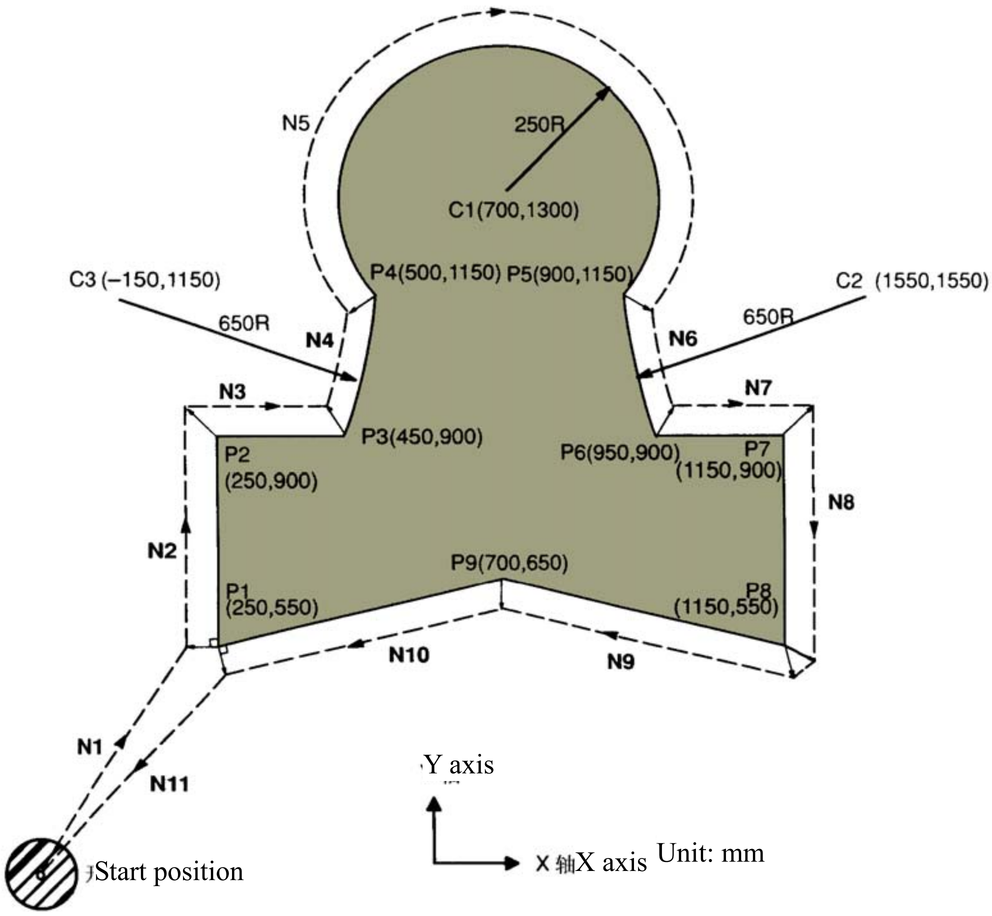
6.1.5 Caution

- In initial status CNC is in cutter radius compensation cancellation mode. CNC sets cutter radius compensation offset mode when the G41 or G42 command is executed. At the beginning of the compensation, the CNC reads two blocks in advance, the next block is stored in the cutter radius compensation buffer memory when a block is performed. When in Single mode, two blocks are read, after the end point of the 1st block is performed, it is stopped. Two blocks are read in advance in successive performance. So, there are a block being performed and two blocks behind it in CNC.
- Neither setup nor cancellation of the Tool compensation C can be performed in the MDI mode.
- The cutter radius compensation value can not be a negative, normally, the wearing value is negative (negative value indicates for wearing)
- Instead of G02 or G03, the setting or cancellation of cutter radius compensation can be commanded only by using G00 or G01, or the alarm occurs.
- CNC will cancel Tool compensation C mode when you press RESET key.
- Corresponding offset should be specified while the G40, G41 or G42 is specified in the block, or the alarm occurs.
- When cutter radius compensation is employed in main program and subprogram, the CNC should cancel compensation mode before calling or exiting sub-program (namely, before M98 or M99 is performed), or the alarm occurs.

Cancel the compensation mode temporarily when G54-59, G28-31 and canned cycle command are executed. Restore the cutter radius compensation mode when the above commands are finished.

6.1.6 Example for application

The parts are machined in the coordinate system in Fig. 6-3. The tool compensation number D07 is employed, tool geometric radius is 2mm and the tool radius wearing is 0.



Perform tool setting in the mode of offset cancellation, after finishing the tool setting, and set the tool radius D in OFFSET page.

Table.4-2

NO.	Geometric(H)	Wearing(H)	Geometric(D)	Wearing(D)
01	…	…	…	…
…	…	…	…	…
07	…	…	2.000	0.000
08	…	…	…	…
…	…	…	…	…
32	…	…	…	…

Programs:

N0 G92 X0 Y0 Z0; Tool are positioned at start position X0, Y0 and Z0 when the absolute coordinate system is specified

N1 G90 G17 G00 G41 D07 X250.0 Y550.0; Start-up cutter, the tool is shifted to the tool path by the distance specified in D07, geometric radius of D07 is set to 2.0mm, tool wearing 0, then the tool radius is 2mm.

N2 G01 Y900.0 F150; Specifies machining from P1 to P2

N3 X450.0; Specifies machining from P2 to P3

N4 G03 X500.0 Y1150.0 R650.0; Specifies machining from P3 to P4

N5 G02 X900.0 R-250.0; Specifies machining from P4 to P5

N6 G03 X950.0 Y900.0 R650.0; Specifies machining from P5 to P6

N7 G01 X1150.0; Specifies machining from P6 to P7

N8 Y550.0; Specifies machining from P7 to P8

N9 X700.0 Y650.0; Specifies machining from P8 to P9

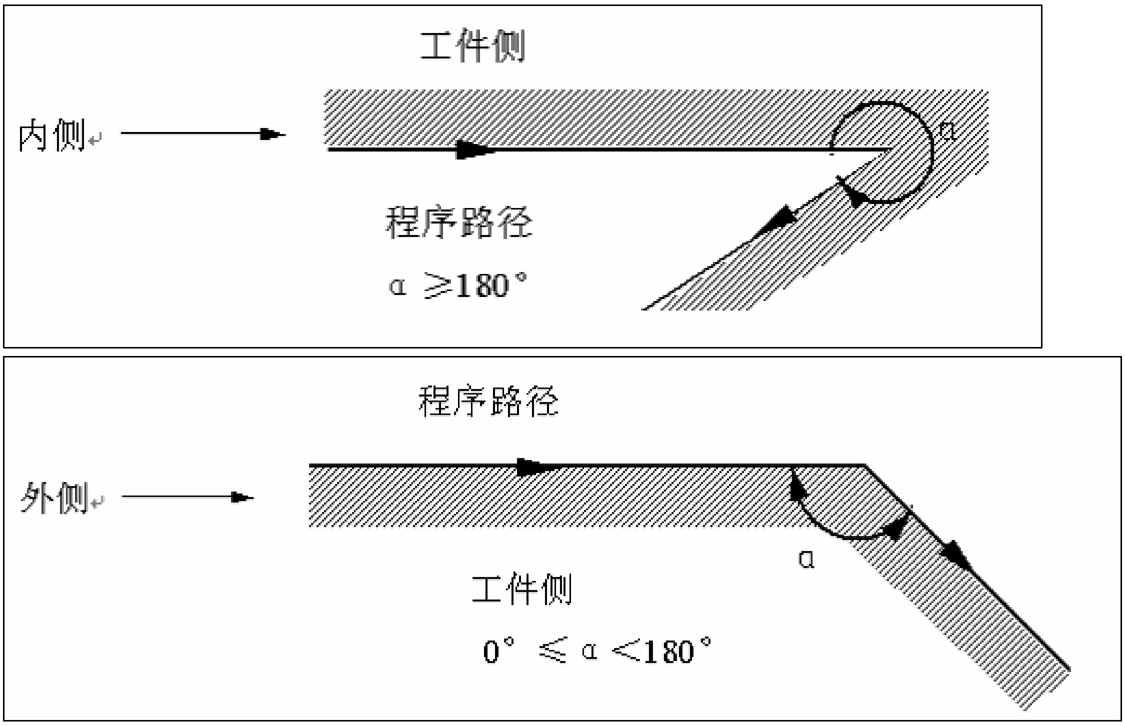
N10 X250.0 Y550.0; Specifies machining from P9 to P1

N11 G00 G40 X0 Y0; Cancels the offset mode, the tool is returned to the start position (X0, Y0)

6.2 Offset Path Explanation for Cutter Radius Compensation

6.2.1 Conception for inner side or outer side

"Inner side" and "outer side" will be employed in the following explanations. When an angle of intersection created by tool paths specified by move commands for two blocks is over or equal to 180°, it is referred to as "inner side". When the angle is between 0° and 180°, it is referred to as "outer side".



6.2.2 Tool movement in start-up

There are 3 steps should be performed for cutter radius compensation: establishment, performing and cancellation.

The tool movement performed from offset cancellation mode to G41 or G42 command establishment is called tool compensation establishment (also called start-up)

Note For S, L and C labeled in the following figures, if not especially described, they should be regarded as the following meaning:

S----Single block stop point;

L----Linear;

C---Circular arc.

(a) Tool movement along an inner side of a corner（α≥180°）

1）Linear to linear

2）Linear to circular

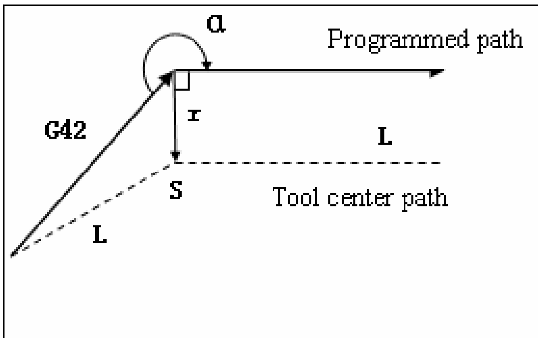


Fig.6-4a Linear to linear (start-up from inner side)

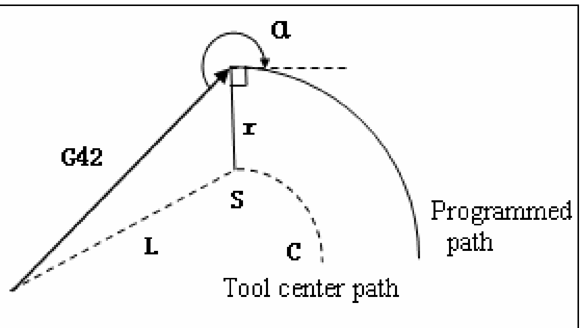


Fig.6-4b Linear to circular (start-up from inner side)

(b) Tool movement along the outside of a corner at an obtuse angle（180°＞α≥90°）

1）Linear to linear

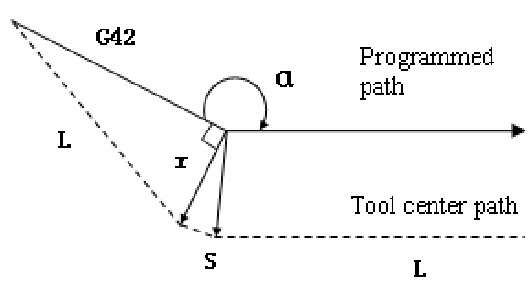


Fig.6-5a Linear to linear (start-up outside)

2) Linear to linear

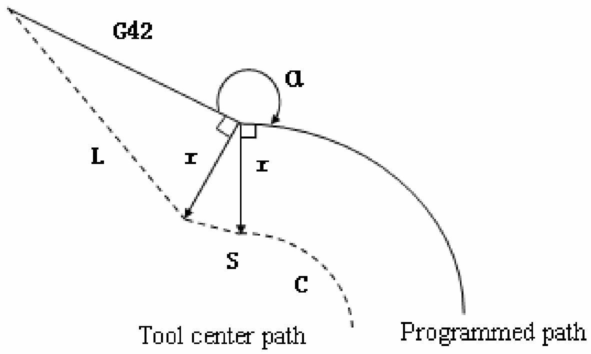


Fig.6-5b Linear to circular (Start-up outside)

(c) Tool movement along the outer side of a corner at an actuate angle（α＜90°）

1）Linear to Linear

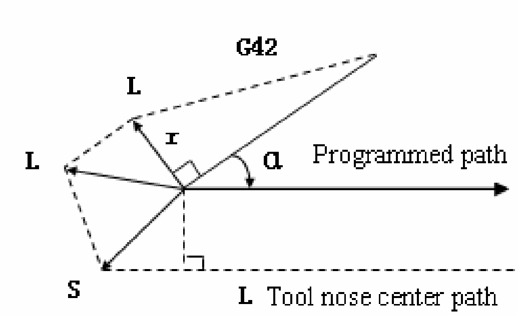


Fig.6-6a Linear to linear (start-up from outer side)

2）Linear to circular

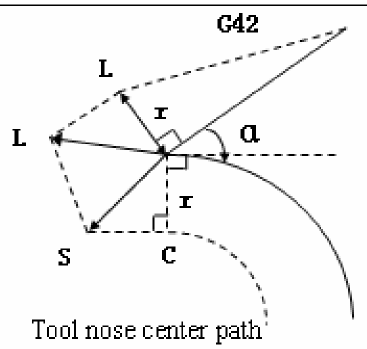


Fig.6-6b Linear to circular (start-up from outer side)

(d) Tool movement along the outside linear to linear at an acute angle less than 1 degree（α≦1°）

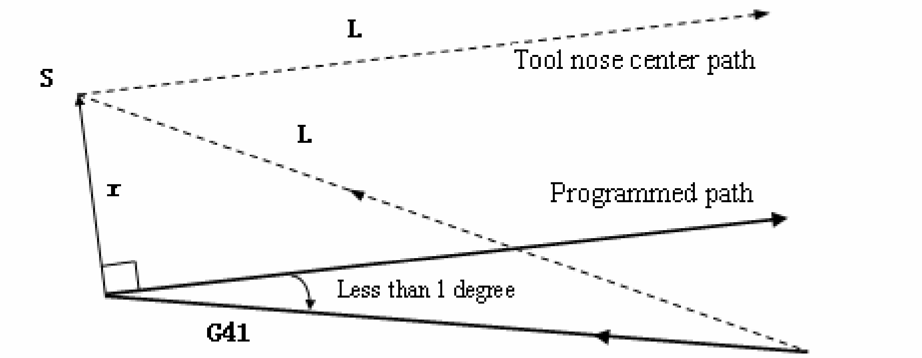


Fig.6-7 Linear to linear (the corner is less than 1 degree, start-up from outer side)

6.2.3 Tool movement in offset mode

The mode after setting the cutter radius compensation and before canceling the cutter radius compensation is called offset mode.

- **Offset path of invariable compensation direction in compensation mode**

1）Linear to linear　　2）Linear to circular

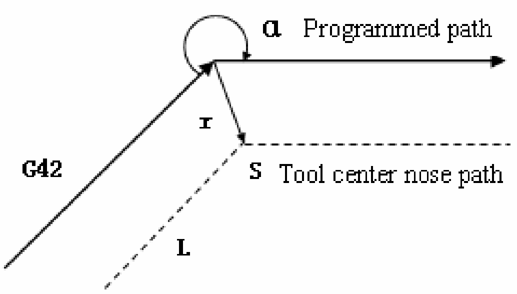


Fig.6-8a Linear to linear (inside movement)

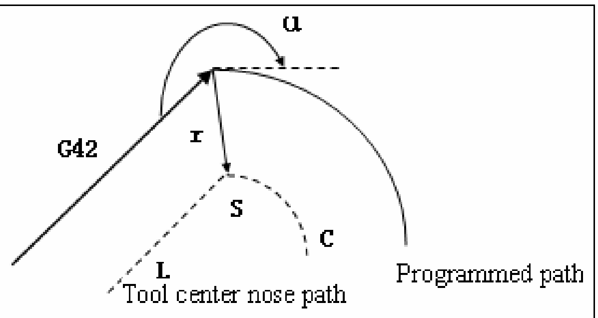


Fig.6-8b Linear to circular (inside movement)

3) Circular to linear

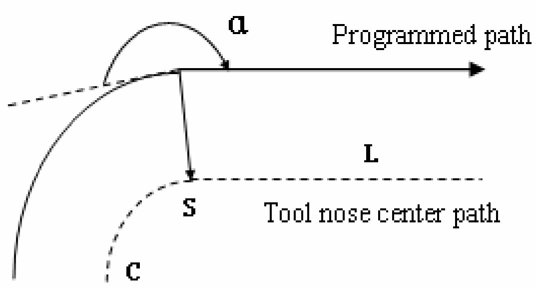


Fig.6-8c Circular to linear (inside movement)

4) Circular to circular

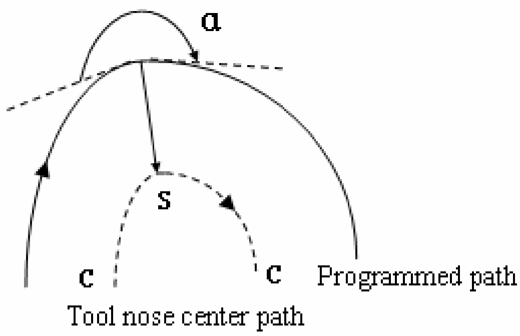


Fig.6-8d Circular to circular (inside movement)

5）Inner side machining less than 1 degree and compensation vector amplification

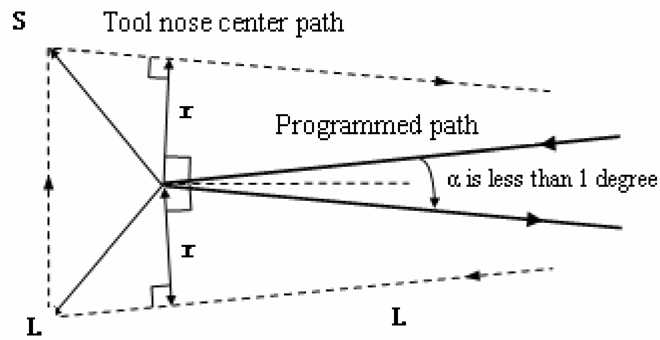


Fig.6-8e Linear to linear (corner is less than 1 degree, inside movement)

（b） Move along the outer of abtuse angle corner（180°＞α≥90°）

1）Linear to linear

2）Linear to circular

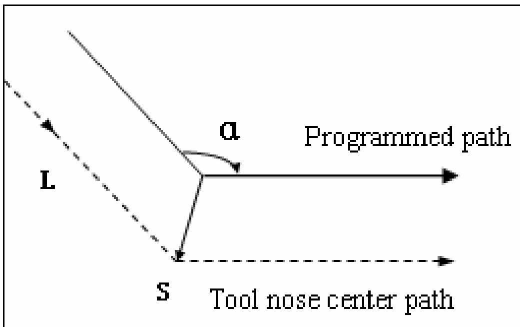


Fig.6-9a Linear to linear
(obtuse angle, outside movement)

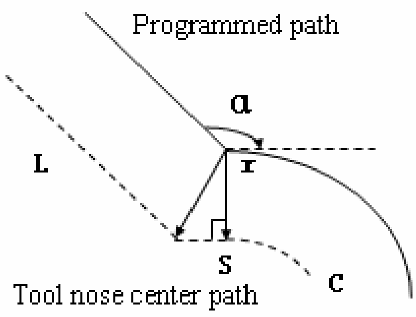


Fig.6-9b Linear to circular
(obtuse, outside movement)

3）Linear to linear

4）Circular to circular

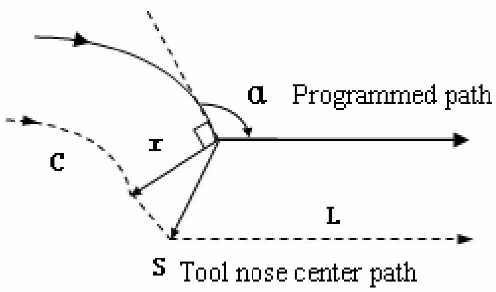


Fig.6-9c Circular to linear
(obtuse angle, outside movement)

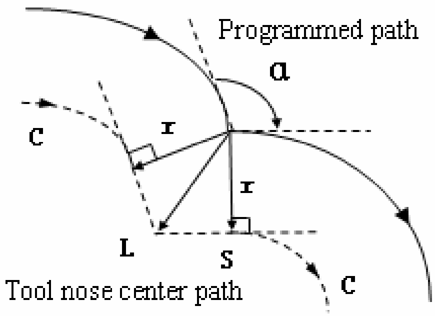


Fig.6-9d Circular to circular
(obtuse angle, outside movement)

（c）Move along the outer of acute angle corner（α<90°）

1）Linear to linear

2）Linear to circular

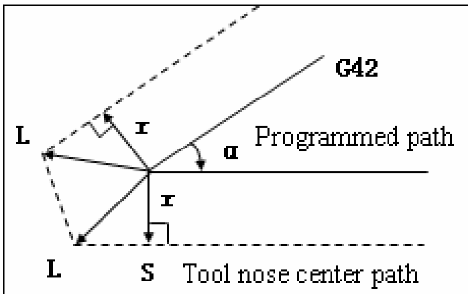


Fig.6-10a Linear to linear
(acute, movement outside)

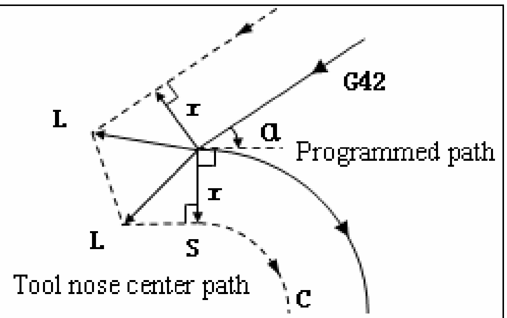


Fig.6-10b Linear to circular
(acute, movement outside)

3）Circular to linear

4)Circular to circular

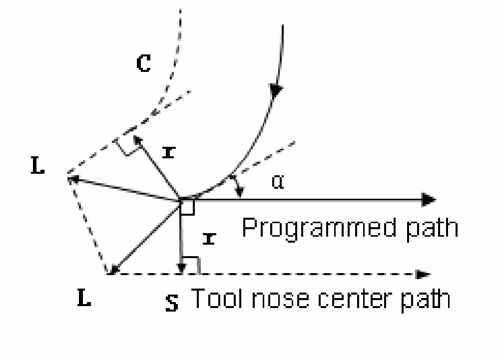


Fig.6-10c Circular to linear (acute, movement outside)

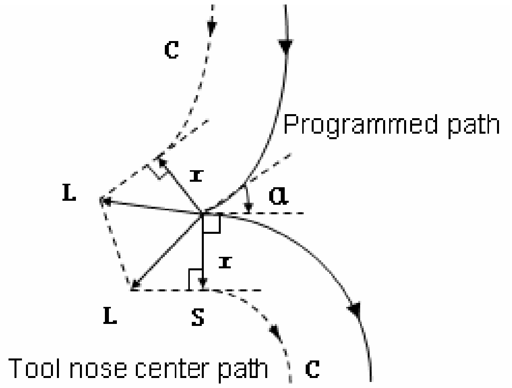


Fig.6-10d Circular to circular (acute, movement outside)

(d) When it is exceptional

1)There is no intersection

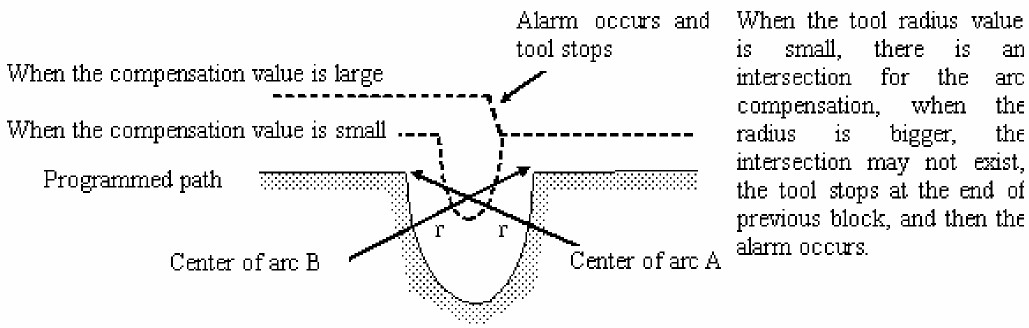


Fig.6-11 Exceptional ------There is no intersection after the path offset

2）The arc center is consistent to the start point or end point

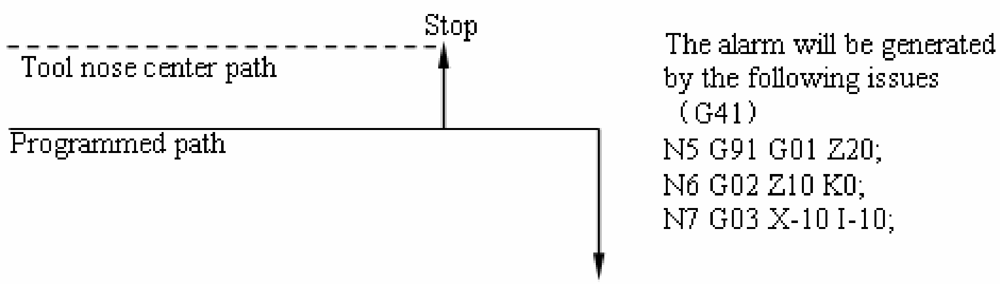


Fig.6-12 Center of arc is consistent to the start point or end point

- **Offset path with the compensation direction changed in compensation mode**

.

The compensation direction can be changed in special occasion, but it cannot be changed at the beginning and the following block. There are no inner side and outer side for the full compensation.

1）Linear to linear

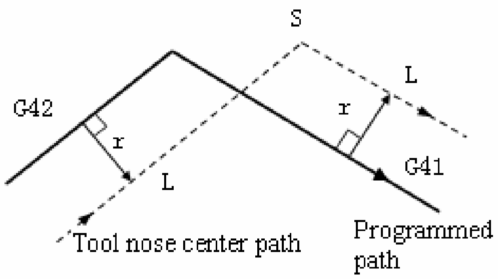


Fig.6-13a Linear to linear (compensation direction changed)

2）Linear to Circular

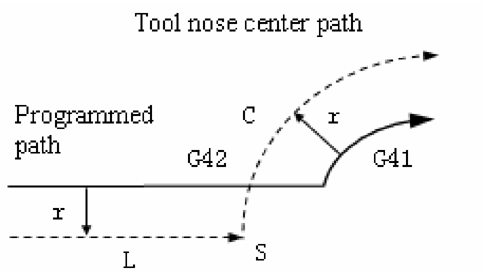


Fig.6-13b Linear to circular (compensation direction changed)

3）Circular to linear

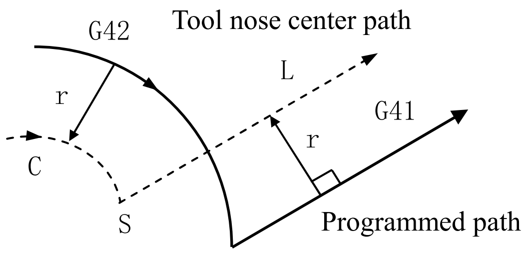


Fig.6-13c Circular to linear (compensation direction changed)

4）Circular to Circular

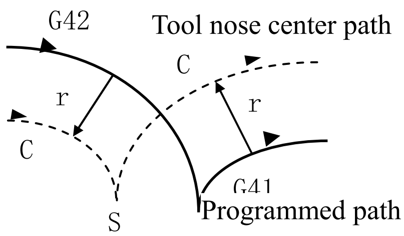


Fig.6-13d Circular to circular (compensation direction changed)

5）When there is no intersection if the compensation is normally performed

When changing the offset direction from block A to block B using G41 and G42, if the intersection of the offset path is not required, create the vector vertical to block B at the start point of block B.

i) Linear to linear

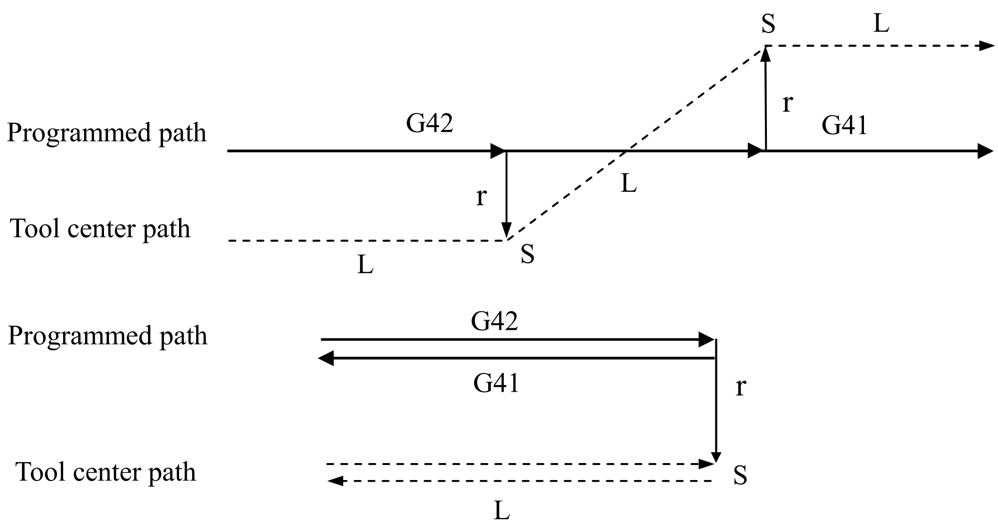


Fig.6-14a Linear to linear, there is no intersection (Compensation direction changed)

ii) Linear to circular

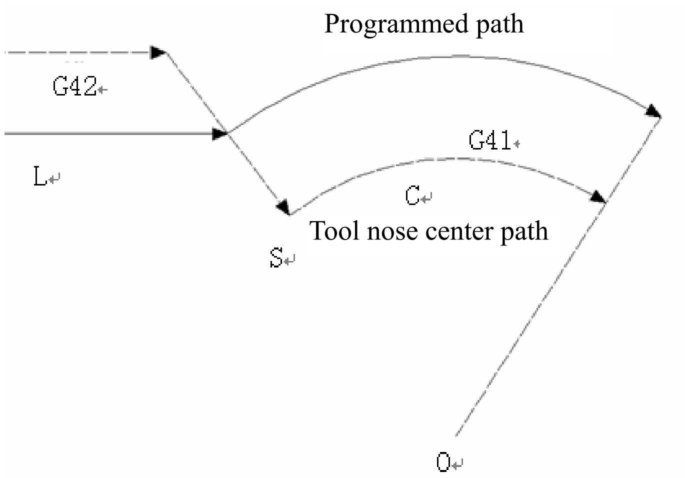


Fig.6-14b Linear to circular, there is no intersection
(Compensation direction changed)

iii) Circular to circular

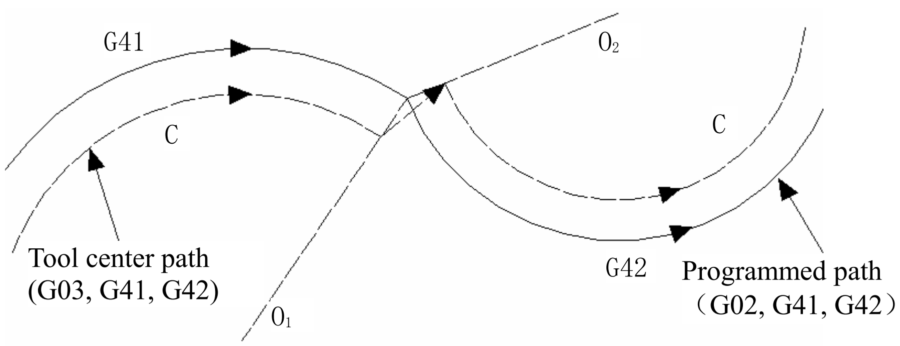


Fig.6-14c Circular to circular, there is no intersection
(Compensation direction changed)

6.2.4 Tool operation in offset cancellation mode

When the G40 command is employed in block in compensation mode, the CNC enters the compensation cancellation mode. This is called compensation cancellation.

The circular arc command (G02 and G03) can not be employed when the cutter radius compensation C is cancelled. If they are commanded, alarm is generated and the operation is stopped

It controls and performs this block and the blocks in the cutter radius compensation buffer memory in the compensation cancellation mode. If the single block switch is turned on, it stops after executing a block. The next block is executed instead of reading it when the start key is pressed again

(a) Tool movement along an inner side of a corner （α≥180°）

1）Linear to linear 2）Circular to linear

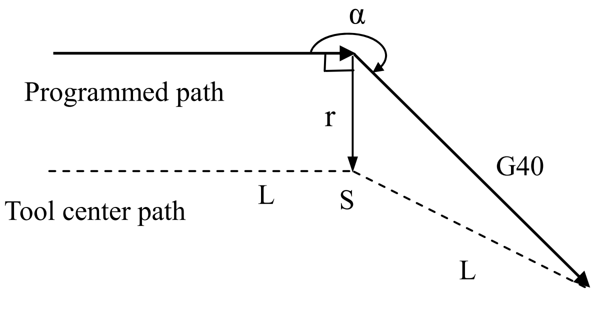


Fig.6-15a Linear to linear
(inner side, offset cancellation)

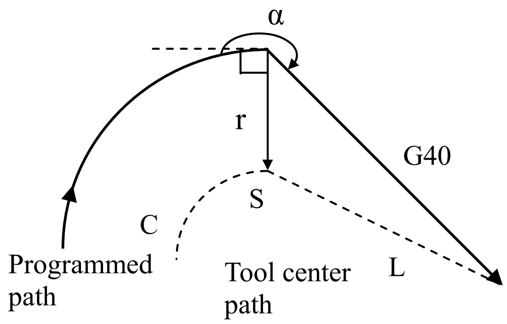


Fig.6-15b Circular to linear
(inner side, offset cancellation)

(b) Tool movement along the outside of a corner at an obtuse angle （180°＞α≥90°）

1）Linear to linear

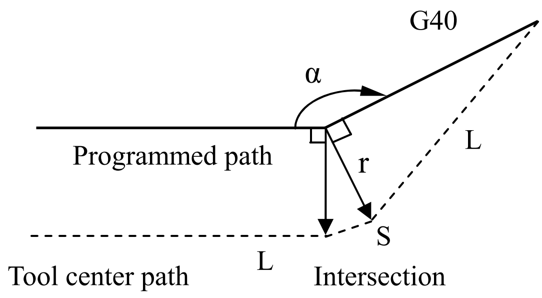


Fig.6-16a Circular to linear
(obtuse, outside, offset cancellation)

2）Circular to linear

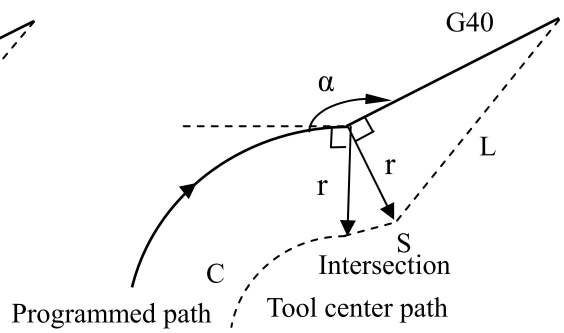


Fig.6-16b Circular to linear
(obtuse, outside, offset cancellation)

(c) Tool movement along the outside of a corner at an acute angle （180°＞α≥90°）

1）Linear to linear

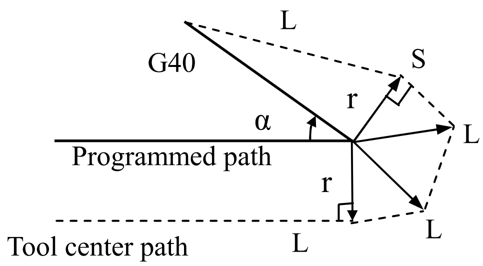


Fig.6-17aLinear to linear
(acute angle, outside, offset cancellation)

2）Circular to linear

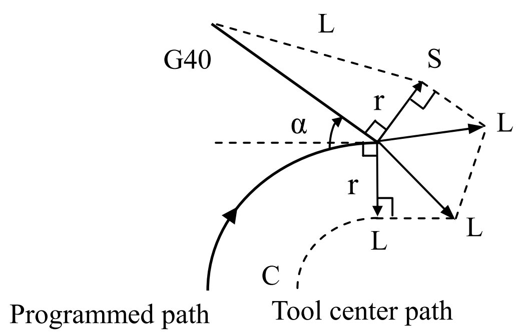


Fig.6-17b Linear to linear
(acute angle, outside, offset cancellation)

(d) Tool movement along the corner outside at an acute angle less than 1 degree: linear to linear（α＜1°

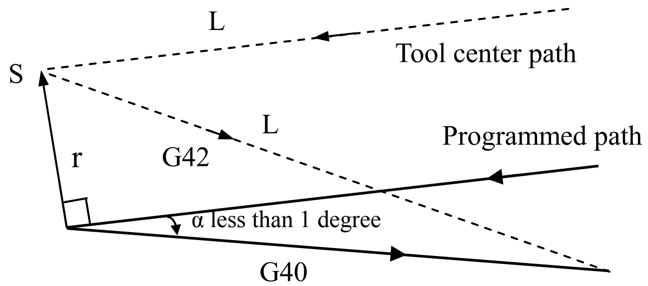


Fig.6-18 Linear to linear (the included angle less than 1 degree, outside, offset cancellation)

6.2.5 Interference check

Tool over cutting is called “interference”. The interference check function can check tool over cutting in advance. This interference check is performed even if the over cutting does not occur. However, all interference can not be checked by this function.

(1) Conditions for the interference

1) The direction of the tool path is different from that of the programmed path. (90 degrees to 270 degrees between these paths)

2) In addition to the condition above, the angle between the start point and end point of the tool center path is quite different from that between the start point and end point of the programmed path in circular machining (more than 180 degrees).

Example: Linear machining

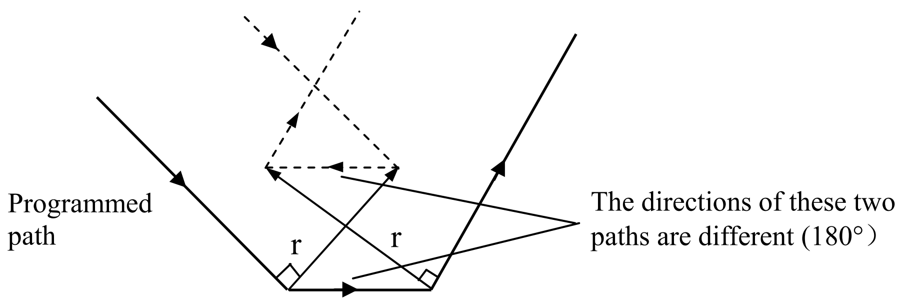


Fig.6-19a Machining interference (1)

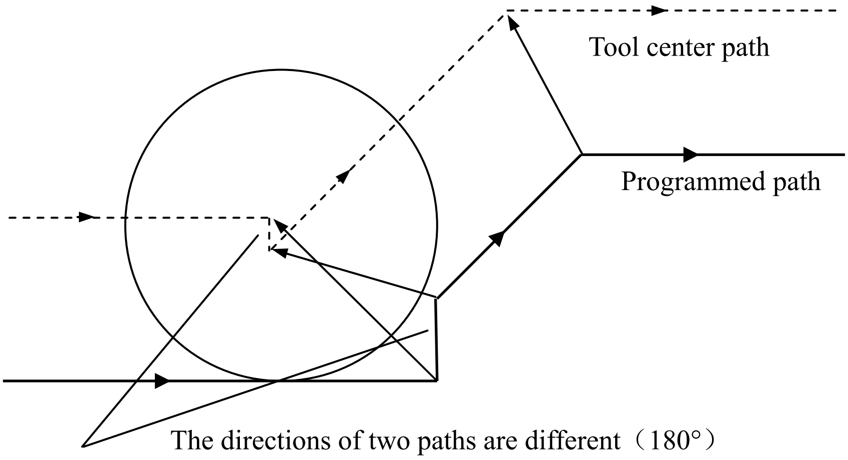


Fig.6-19b Machining interference (2)

(2) If there is no interference actually, but it is treated as interference.

1) The groove depth less than the compensation value

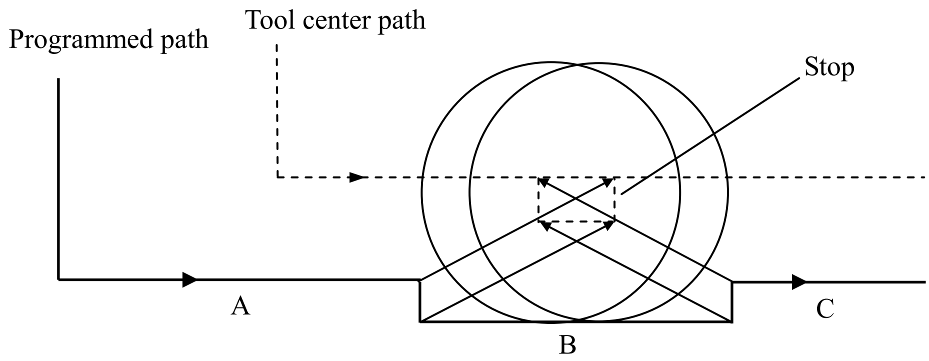


Fig.6-20 Exceptional case (1) treated as interference

There is no interference actually, but program direction in block B is opposite to the cutter radius compensation path. The cutter stops, and the alarm occurs.

2) The groove depth less than compensation value

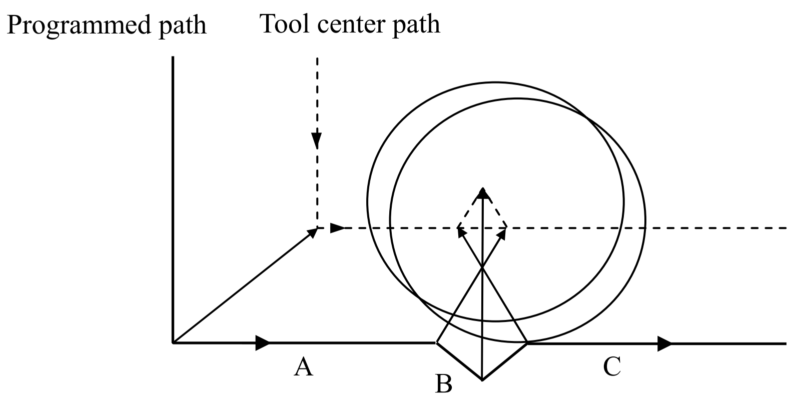


Fig.6-21 Exceptional case (2) treated as interference

There is no interference actually, but program direction in block B is opposite to the cutter radius compensation path. The cutter stops, and the alarm occurs.

6.2.6 Command of compensation vector cancel temporarily

If the following commands G92, G28, G29, coordinate command selection G54~G59 and canned cycle are specified in compensation mode, the compensation vector is temporarily cancelled and then automatically restored after these commands are executed. Now, the temporary compensation vector cancellation is different to the compensation cancellation mode, tool is moved to the specified point by compensation vector cancellation from the intersection. And the tool moves to the intersection directly when the compensation mode restores.

- **Coordinate system setting command G92 and coordinate system selection command G54~G59**

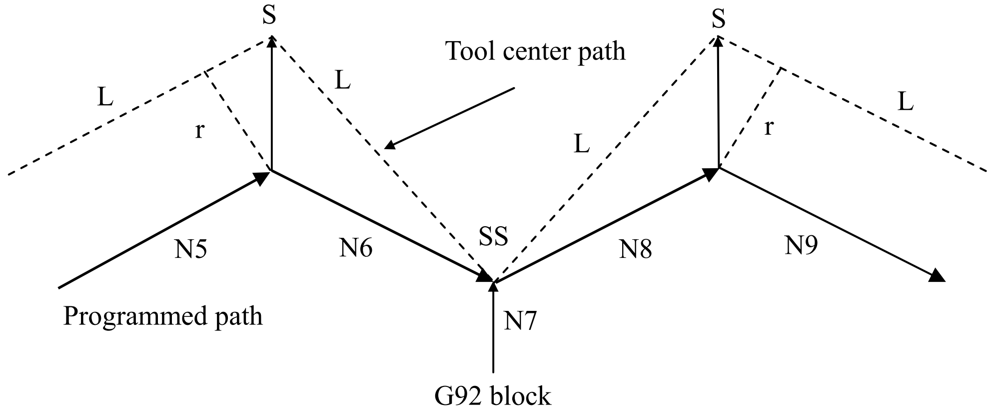


Fig.6-22 Temporary compensation vector by G92

Note: SS is indicated as the point stopped for twice in Single block mode.

- **Automatic return to the reference point G28**

If G28 is specified in compensation mode, the compensation will be cancelled at an intermediate position. The compensation mode is automatically restored after the reference point is returned.

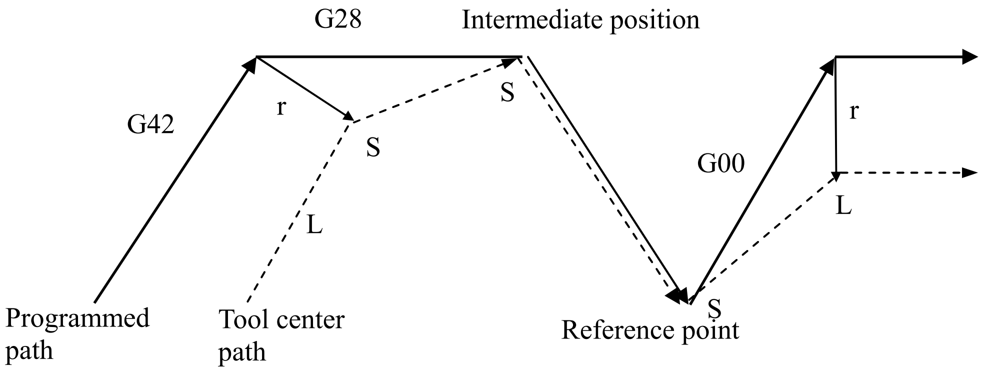


Fig.6-23 Temporarily cancel compensation vector by G28

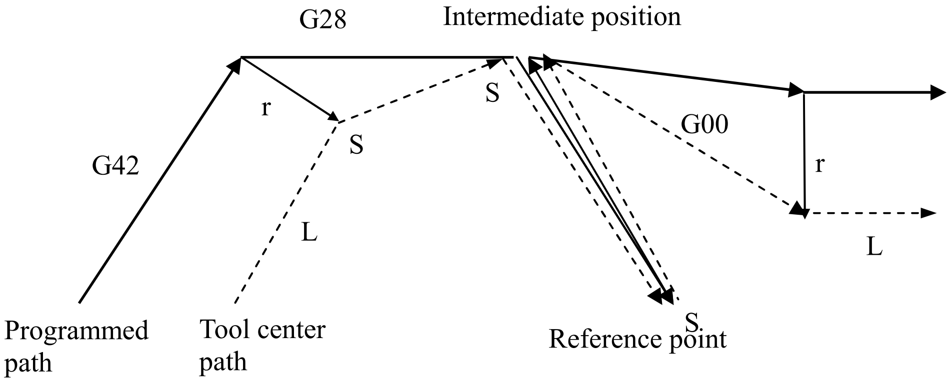


Fig. 6-24 G29 temporarily cancel compensation vector

- **Canned cycle**

If the canned cycle command is specified in compensation mode, the compensation will be temporarily cancelled in the canned cycle operation 1. The compensation mode is automatically restored after the canned cycle is terminated.

6.2.7 Exceptional case

- **When the inner corner machining is less than tool radius**

When the inner corner machining is less than tool radius, the inner offset of a tool will cause over cut. The tool stops and alarm occurs after moving at the beginning or at the corner in previous block. But if the switch of "Single block" is ON, the tool will be stopped at the end of the previous block.

- **When a groove less than the tool diameter is machined**

When the tool center moves opposite to the direction of programmed path, the over cutting will be generated by the cutter radius compensation. Tool stops and alarm appears after moving at the beginning of previous block or at the corner.

- **When a step less than the tool radius is machined**

When a program contains a step which is an arc and less than tool radius, tool center path may form a opposite movement direction to the programmed path. So the first vector is ignored and it moves to the end of the second vector along a straight line. The program will be stopped for Single block mode, the cycle continues if it is not single block mode. The compensation will be executed correctly and no alarm will be generated if the step is a straight line. (But the uncut part is reserved.)

- **When the sub-program is contained in G code**

CNC should be in compensation cancellation mode before calling the sub-program (namely, before the G98 is performed). Offset can be applied after entering the sub-program, but the compensation cancellation should be applied before returning to the main-program (before M99), or the alarm occurs.

- **When compensation value is changed**

(a) Usually, the compensation value is changed when the tool change is performed in compensation cancellation mode. If the compensation value is changed in compensation mode, the

new one is ineffective which is effective till the program is executed again.

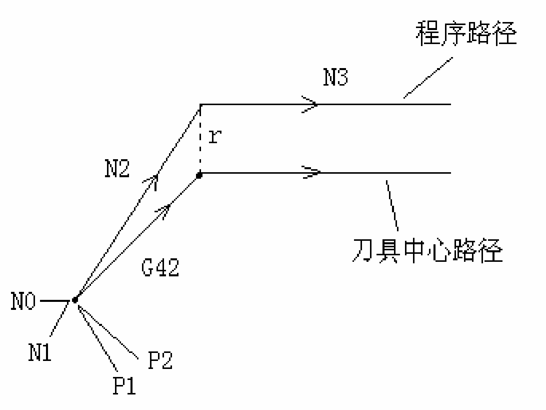
(b) If different compensation values are commanded in different blocks of a program, different compensation value will be compensated to the corresponding block. But if it is an arc, the alarm will be generated. For details, refer to the following explanation.

(c) about “arc data error in C type cutter radius compensation”.

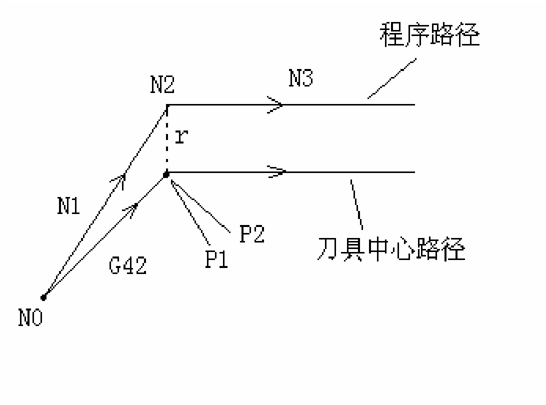
- **When the end point for the programming arc is not on the arc**

When the end point for the programming arc is not on the arc, the tool stops and the alarm information shows “end point is not on the arc”.

Two same points in the starting is shown an example:



N0 G90 G00 X-50 Y-50
N1 G91 G1 G41 X0 Y0 D1 F800 …without moving
N2 G90 X0 Y0
N3 X50



The above-mentioned program may occur the “two same points” when starting, and the compensation may not perform. The transit point P1 between N0 and N1 and the transit point P2 between N1 and N2 are shared a same point.

N0 G90 G00 X-50 Y-50
N1 G1 G41 X0 Y0 D1 F800
N2 G91 X0 Y0 …without moving
N3 X50

The “last two same points” may occur when starting at the last program, in the case of the compensation has been performed. The section without moving which is regarded as the movement

approximates to the zero, so it is necessary to maintain the compensation amount. The transit point between N1 and N2 is P1, and the transit point between N2 and N3 is P2, P1 and P2 are shared a same point.

In the same way, in the compensation mode, if the “two same points” may occur, the compensation value will be maintained; in the retraction mode, the similar start mode is divided into “the previous two same points” and “the last two same points”

- **The alarm and corresponding explanation of ‘Circular arc data error in cutter compensation C’**

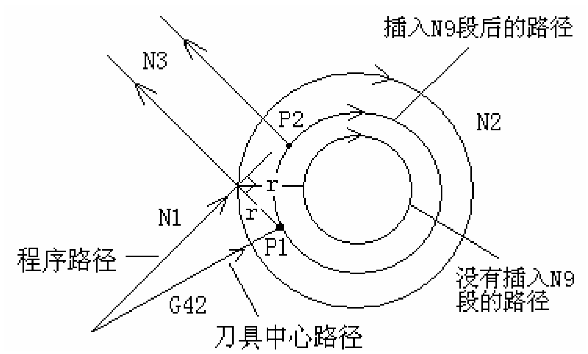
(a) The example of this alarm may occur in a circle

Porgram example：N0 G90 G00 X-50 Y-50 Z50
N1 G01 G42 X0 Y0 D1 F800
N2 G02 I50
N3 G91 G01 X-50 Y-50

程序路径：Programmed path

刀具中心路径：Tool center path

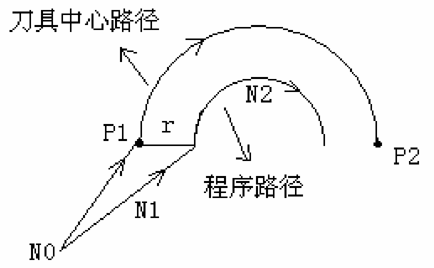
The transit point between straight line N1 and circular arc N2 is P1, the transit point between circular N2 and straight line N3 is P2, and the compensation radius is r, in this case, the circular after tool compensation is more than 360°.



After a block (N9 G91 G0 X0 Y0) (without moving) is inserted between N1 and N2 in the above-mentioned program, the “circular data error in cutter compensation C” may alarm.

Because the point after N9 inserted which is equal to the one of N1, namely, they are regarded as “two same points”. The transit point P1 is performed treating the “two same points”, the position of P1 is obviously differ from the above one which does not insert the N9 block. So the cut circular arc path by this transit pont is absolutely differing from the path to be machined, so the alamr is then generated: “circular arc data error in cutter compensation C”

(b) The example for a non-circle may occur:



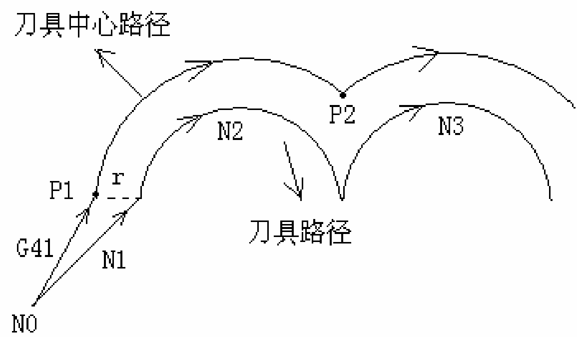
Program example: N0 G90 G00 X-50 Y-50 Z50
N1 G01 G41 X0 Y0 D1 F800
N2 G02 X50 R25

The P1 and P2 are the transit point of tool compensation as the left figure shown, wherein the “r” is compensation radius. This is a normal treatment mode for the straight line to circular arc.

The alarm may occur in terms of the following program
N0 G90 G00 X0 Y0 Z0
N1 G01 G41 X0 Y0 D1 F800 …without moving, originally start
N2 G02 X50 R25

Because the N1 block does not a movement, namaly, it equals to the “two same points”. The transit points P1 and P2 are performed based on the treatment of two same points (The path of two same points), so the circular arc path cut by this transit point obviously differs from the actual path to be machined, in this case, the “circular arc data error in cutter compensation C” may alarm.

(c) In the calculation of arc cutter compensation C, this alarm may issue if the compensation radius D is modified.



Program example: N0 G90 G00 X-50 Y-50 Z25
N1 G01 G41 X0 Y0 D1 F800
N2 G02 X50 R25
N3 G02 X100 R25

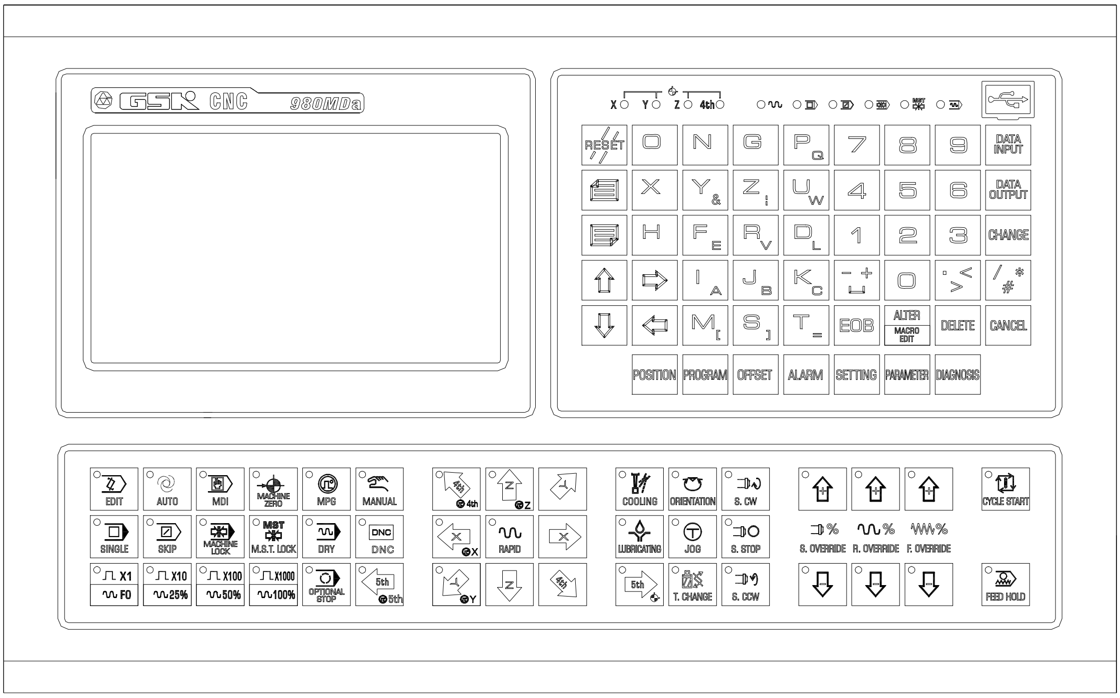
The left figure is shown the programmed path and the tool center path.

If the compensation radius D is changed in N3, for example, the D2 is speicified in N3 block (the value of D2 is not equal to the one of D1), in this case, it is similar as (b), an alarm of the “circular arc data error in cutter compensation C” may occur.

VOLUME Ⅱ OPERATION

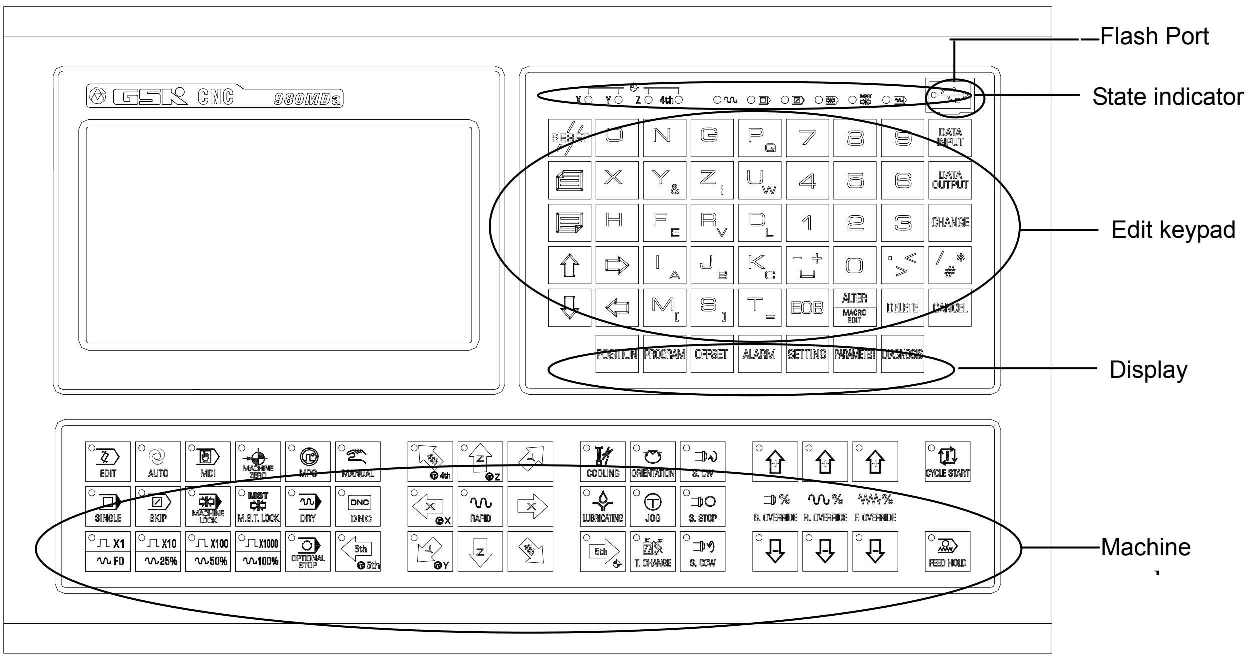
CHAPTER1 OPERATION MODE AND DISPLAY

This GSK980MDa system employs an aluminum alloy solid operator panel, which exterior is as follows.

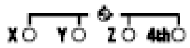
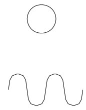
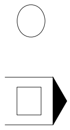
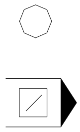
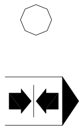
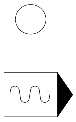


1.1 Panel Division

This GSK980MDa adopts an integrated panel, which division is as follows:

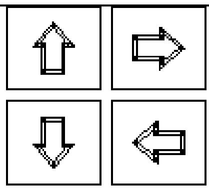
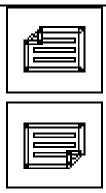


1.1.1 State indication

	machine zero return finish indicator		Rapid indicator
	Single block indicator		Block Skip indicator
	Machine Lock indicator		MST Lock indicator
	Dry Run indicator		

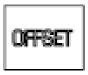
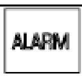
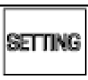
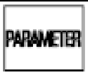
1.1.2 Edit keypad

Key	Name	Function
RESET	RESET key	For CNC reset, feed, output stop etc.
O, N, G, P/Q, X, Y/&, Z/!, U/W, H, F/E, R/V, D/L	Address key	Address input
I/A, J/B, K/C, M/[, S/], T/=		Double address key, switching between two sides by pressing repeatedly
- + ␣, / * #	Sign key	Double address key, switching between two characters by pressing repeatedly

Key	Name	Function
7 8 9 4 5 6 1 2 3 0	Numerical key	For digit input
. < >	Decimal point	For decimal point input
DATA INPUT	Input key	For confirmation of parameters, offset values input
DATA OUTPUT	Output key	For start communication output
CHANGE	Change key	For switching of message, display
ALTER/MACRO EDIT DELETE CANCEL	Edit key	For insertion, alteration, deletion of programs, words in editing(ALTER/MACRO EDIT is a compound key, switching between two functions by pressing repeatedly)
EOB	EOB key	For block end sign input
	Cursor moving keys	For cursor moving control
	Page key	Page switching in a same interface

1.1.3 Menu display

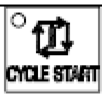
Menu key	Remark
POSITION	**To enter position interface. There are RELATIVE POS, ABSOLUTE POS, INTEGRATED POS, POS&PRG pages in this interface.**

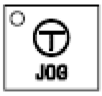
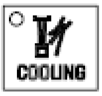
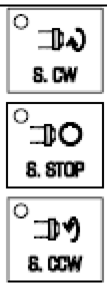
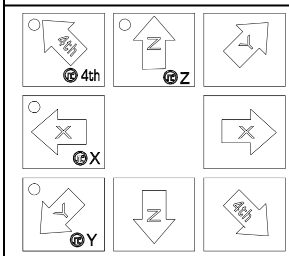
	**To enter program interface. There are PRG CONTENT, PRG STATE, PRG LIST, PRG PREVIEW,4 pages in this interface.**
	**To enter TOOL OFFSET interface. There are TOOL OFFSET, MARRO variables and Tool Life Management (modifying Bit0 of state parameter №002). OFFSET interface displays offset values; MARRO for CNC macro variables.**
	**To enter alarm interface. There are CNC, PLC ALARM and ALARM Log pages in this interface.**
	**To enter Setting interface. There are SWITCH, PASSWORD SETTING, DATE &TIME, SETTING （G54～G59）, GRAGH SET and TRACK pages in this interface.**
	**To enter BIT PARAMETER, DATA PARAMETER, PITCH COMP interfaces (switching between each interface by pressing repeatedly).**
	**To enter DIAGNOSIS interface.There are CNC DIAGNOSIS, PLC STATE, PLC VALUE, VERSION MESSAGE interfaces (switching between each interfaces by pressing the key repeatedly). CNC DIAGNOSIS, PLC STATE, PLC VALUE interfaces display CNC internal signal state, PLC addresses, data state message; the VERSION MESSAGE interface displays CNC software, hardware and PLC version No.**

1.1.4 Machine panel

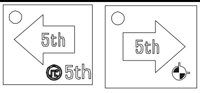
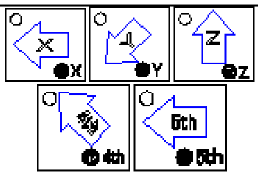
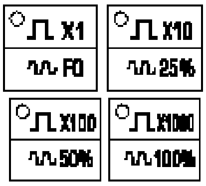
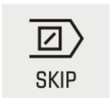
The keys function in GSK980MDa machine panel is defined by PLC program (ladder), see their function significance in the machine builder's manual.

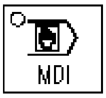
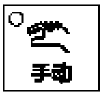
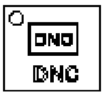
The functions of the machine panel keys defined by standard PLC program are as follows:

Key	Name	Function explanation	Function mode
	Feed Hold key	Dwell commanded by program, MDI	Auto mode, DNC, MDI mode
	Cycle Start key	Cycle start commanded by program, MDI	Auto mode, DNC, MDI mode
	Feedrate Override keys	For adjustment of the feedrate	Auto mode, DNC, MDI mode, Edit mode, Machine zero mode, MPG mode, Single Step mode, MANUAL mode

Key	Name	Function explanation	Function mode
	Rapid override keys	For adjustment of rapid traverse	Auto mode, DNC, MDI mode, Machine zero mode, MANUAL mode
	Spindle override keys	For spindle speed adjustment (spindle analog control valid)	Auto mode, DNC, MDI mode,edit mode, Machine zero mode, MPG mode, Step mode, MANUAL mode
	JOG key	For spindle Jog ON/OFF	Machine zero mode, MPG mode, Single Step mode, MANUAL mode,
	Lubricating key	For machine lubrication ON/OFF	Machine zero mode, MPGmode, Single Step mode,MANUAL mode,
	Cooling key	For coolant ON/OFF	Auto mode, MDI mode,Edit mode, Machine zero mode, MPG mode Step mode, MANUAL mode
	Spindle control keys	Spindle CCW Spindle stop Spindle CW	Machine zero mode, MPGmode, Single Step mode,MANUAL mode,
	Rapid traverse key	For rapid traverse /feedrate switching	Auto mode, DNC,MDI mode, Machine zero mode, MANUAL mode,
	Manual feed key	For positive/negative moving of X, Y, Z axis in Manual, Step mode	Machine zero mode, Step mode, MANUAL mode,

Volume II Operation

Key	Name	Function explanation	Function mode
			
	Handwheel axis selection key	For X, Y, Z axis selection in MPG mode	MPG mode
	MPG/Step increment and Rapid override selection key	Move amount per handwheel scale 0.001/0.01/0.1 mm Move amount per step 0.001/0.01/0.1 mm	Auto mode, MDI mode, Machine zero mode, MPG mode, Step mode,MANUAL mode,
	Single Block key	For switching of block/blocks execution, Single block lamp lights up if Single mode is valid	Auto mode, DNC, MDI mode
	Block Skip key	For skipping of block headed with"/"sign, if its switch is set for ON, the Block Skip indicator lights up	Auto mode, DNC, MDI mode
	Machine Lock key	If the machine is locked, its lamp lights up, and X, Z axis output is invalid.	Auto mode, DNC, MDI mode, Edit mode, Machine zero mode, MPG mode, Step mode, MANUAL mode,
	M.S.T. Lock key	If the miscellaneous function is locked, its lamp lights up and M, S, T function output is invalid.	Auto mode, DNC, MDI mode
	Dry Run key	If dry run is valid, the Dry run lamp lights up. Dry run for program/MDI blocks command	Auto mode, DNC, MDI mode

Key	Name	Function explanation	Function mode
	Edit mode key	To enter Edit mode	Auto mode, DNC, MDI mode, Machine zero mode, MPG mode, Step mode, MANUAL mode
	Auto mode key	To enter Auto mode	MDI mode, DNC, Edit mode, Machine zero mode, MPG mode, Step mode, MANUAL mode,
	MDI mode key	To enter MDI mode	Auto mode, DNC, Edit mode, Machine zero mode, MPG mode, Step mode, MANUAL mode,
	Machine zero mode key	To enter Machine zero mode	Auto mode, DNC, Edit mode, Machine zero mode, MPG mode, Step mode, MANUAL mode,
	Step/MPG mode key	To enter Step or MPG mode (one mode is selected by parameter)	Auto mode, DNC, Edit mode, Machine zero mode, MPG mode, Step mode, MANUAL mode,
	Manual mode key	To enter Manual mode	Auto mode, DNC, Edit mode, Machine zero mode, MPG mode, Step mode, MANUAL mode,===========
	DNC mode key	To enter DNC mode	To enter DNC mode by pressing this key in Auto mode

1.2 Summary of Operation Mode

There are 7 modes that include Edit, Auto, DNC， MDI, Machine zero, Step/MPG, Manual, modes in this GSK980MDa.

- **Edit mode**

  In this mode, the operation of part program setting-up, deletion and modification can be performed.

- **Auto mode**

  In this mode, the program is executed automatically.

- **MDI mode**

In this mode, the operation of parameter input, command blocks input and execution can be performed.

- **Machine zero mode**

  In this mode, the operation of X, Y, Z, 4th, 5th axis machine zero return can be performed separately.

- **MPG / Step mode**

  In the Step/MPG feed mode, the moving is performed by an increment selected by CNC system.

- **Manual mode**

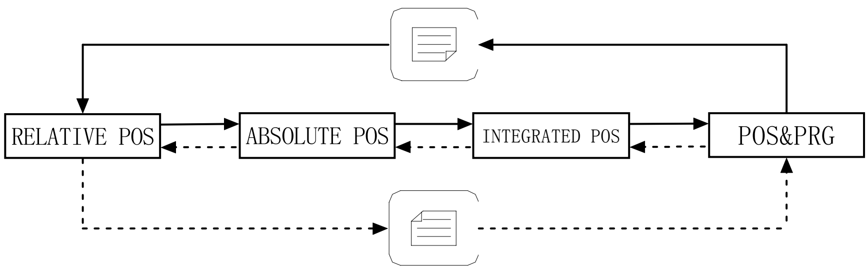
  In this mode, the operation of Manual feed, Manual Rapid, feedrate override adjustment, Rapid override adjustment and spindle ON/OFF, cooling ON/OFF, Lubrication ON/OFF, spindle jog, manual tool change can be performed.

- **DNC mode**

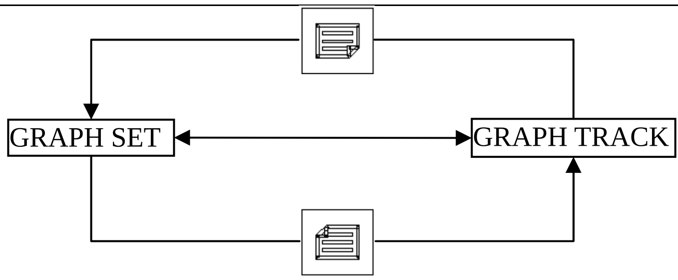
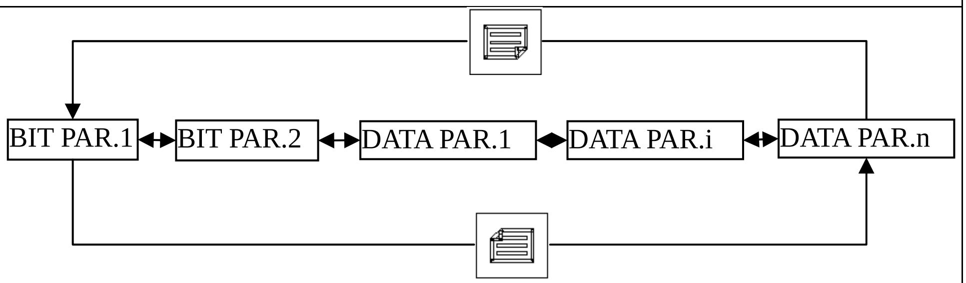
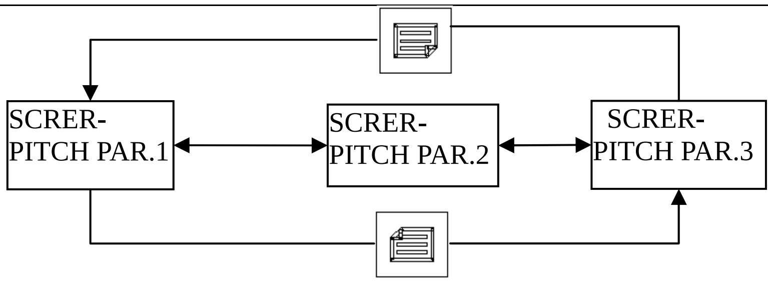
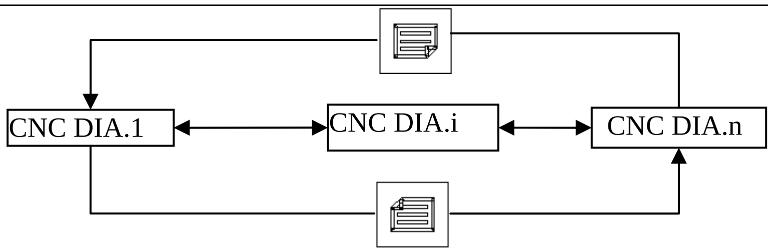
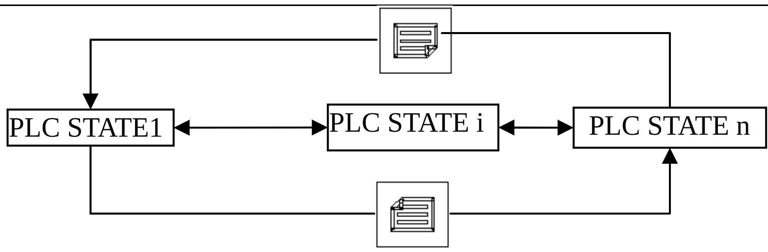
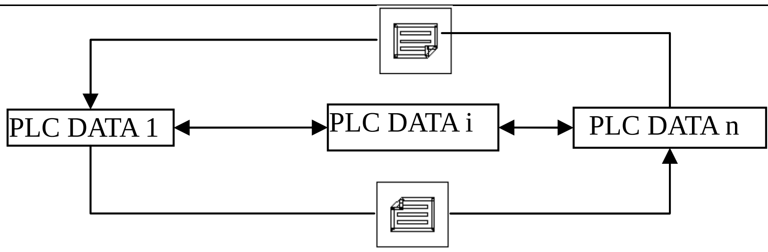
  In this mode, the program is run by DNC mode.

1.3 Display Interface

There are 7 interfaces for GSK980MDa such as Position, Program etc., and there are multiple pages in each interface. Each interface (page) is separated from the operation mode. See the following figures for the display menu, display interface and page layers:

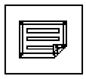
Menu key	Display interface	Display page
POSITION	Position interface	
PROGRAM	Pro. content	PRG CONTENT
	Pro. state	PRG STATE
	Pro.preview	PRG PREVIEW
	Program list	PRG LIST

Menu key	Display interface	Display page
OFFSET	TOOL OFFSET interface	Tool Offset 1 ↔ Tool Offset i ↔ Tool Offset 5
	MACRO interface	MACRO 1 ↔ MACRO i ↔ MACRO 4
	Tool life interface	Tool Life 1 ↔ Tool Life i ↔ Tool Life n
ALARM	CNC alarm	CNC ALARM
	PLC alarm/warn	PLC ALARM/WARN
	Alarm log	ALARM LOG
SETTING	Setting interface	SWITCH SETTING ↔ Time &DATE ↔ AUTH.OPERATION
	G54 setting	SET (G54~G59)

<table>
<tr><th>Menu key</th><th>Display interface</th><th>Display page</th></tr>
<tr><td></td><td>Graph interface</td><td>
</td></tr>
<tr><td rowspan="3">PARAMETER</td><td>Bit parameter</td><td rowspan="2">
</td></tr>
<tr><td>Data parameter</td></tr>
<tr><td>Pitch parameter</td><td>
</td></tr>
<tr><td rowspan="4">DIAGNOSIS</td><td>CNC diagnosis</td><td>
</td></tr>
<tr><td>PLC state</td><td>
</td></tr>
<tr><td>PLC data</td><td>
</td></tr>
<tr><td>Version message</td><td>VERSION MESSAGE</td></tr>
</table>

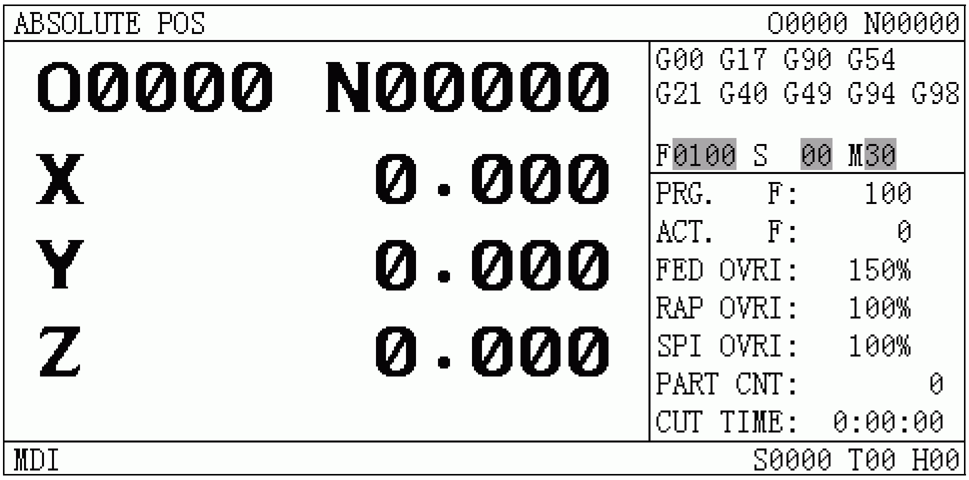
1.3.1 Position interface

Press 

to enter Position interface, which has four interfaces such as ABSOLUTE POS, RELATIVE POS, INTEGRATED POS and POS&PRG, and they can be viewed by  or  key.

1) ABSOLUTE POS display interface

The X, Y, Z coordinates displayed are the absolute position of the tool in current workpiece coordinate system, as CNC power on, these coordinates are held on and the workpiece coordinate system is specified by G92.



PRG. F: a rate specified by F code in program

Note: It displays “PRG. F” in Auto, MDI mode; “MAN. F” in Machine zero, Manual mode;“HNDL INC”in MPG mode; “STEP INC”in Step mode.

ACT. F: Actual speed after feedrate override calculated.

FED OVRI: An override that is selected by feedrate override switch.

SPI OVRI: Adjust the spindle rotational speed by altering spindle override.

PART CNT: Part number plus 1 when M30 (or M99 in the main program) is executed

CUT TIME: Time counting starts if Auto run starts, time units are hour, minute and second

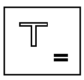
The parts counting and the cut time are memorized at power-down and the clearing ways for them are as follows:

PART CNT clearing: press 

key then press 

key.

CUT TIME clearing: press 

key then press 

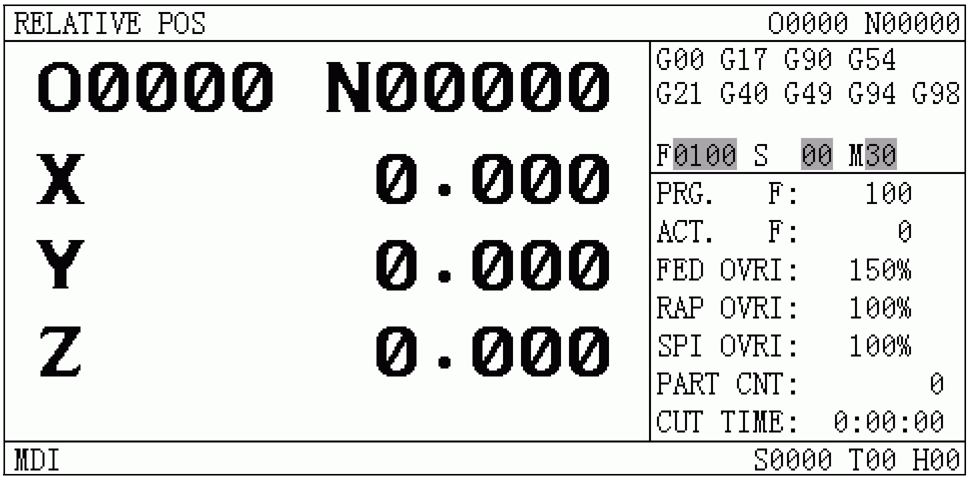
key.

S0000： Feedback spindle speed of spindle encoder, and spindle encoder must be fixed to display actual spindle speed.

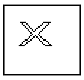
T01： Current tool No. and tool offset No.

2) RELATIVE POS display page

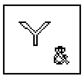
The X, Y, Z axis coordinates displayed are the current position relative to the relative reference point, and they are held on at CNC power on. They can be cleared at any time. If X, Y, Z axis relative coordinates are cleared, the current position will be the relative reference point. When CNC parameter No.005 Bit1=1, as the absolute coordinates are set by G92 code, X, Y, Z axis relative coordinates are identical with the set absolute coordinates.



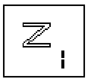
The clearing steps of X, Y, Z axis relative coordinates:

In RELATIVE POS page, press and hold  key till the “X”in the page blinks, press 

key to clear X coordinate;

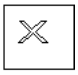
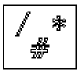
In RELATIVE POS page, press and hold  key till the“Y”in the page blinks, press 

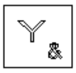
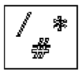
key to clear Y coordinate;

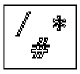
In RELATIVE POS page, press and hold  key till the “Z”in the page blinks, press 

key to clear Z coordinate;

The method for X, Y, Z axis relative coordinates divided by 2:

In RELATIVE POS page, press and hold  key till the “X”in the page blinks, press  key, X coordinate will be divided by 2;

In RELATIVE POS page, press and hold  key till the “Y”in the page blinks, press  key, Y coordinate will be divided by 2;

In RELATIVE POS page, press and hold  key till the “Z”in the page blinks, press  key, Z coordinate will be divided by 2;

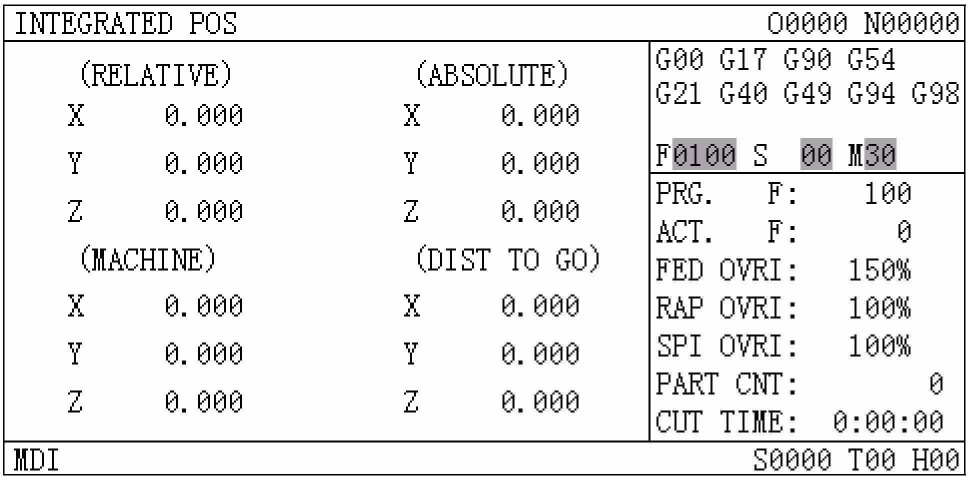
3) INTEGRATED POS display page

In INTEGRATED POS page, the RELATIVE, ABSOLUTE, MACHINE coordinate, DIST TO GO (only in Auto and MDI mode) are displayed together.

The displayed value of MACHINE coordinate is the current position in the machine coordinate system which is set up according to the machine zero.

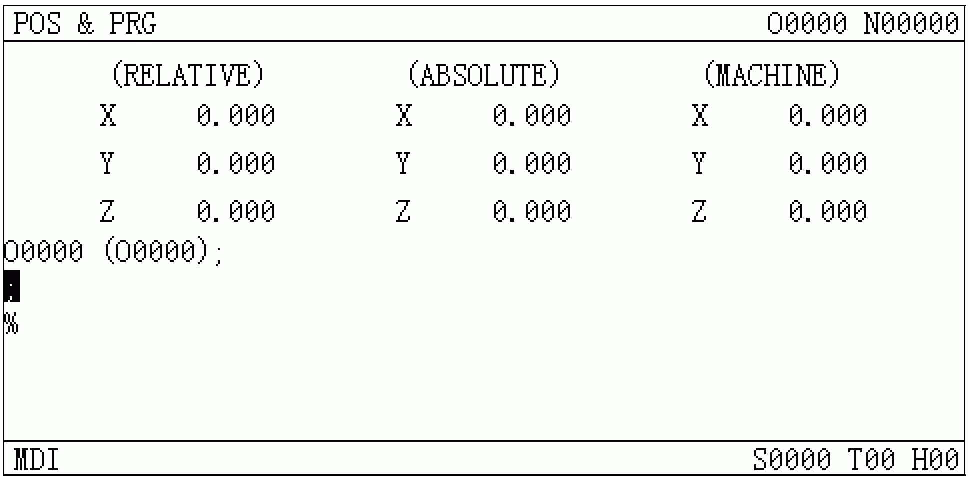
DIST TO GO is the difference between the target position of block or MDI and the current position.

The display page is as follows:



4) POS&PRG display page

In this page, it displays ABSOLUTE, RELATIVE of the current position (ABSOLUTE, DIST TO GO of current position will be displayed if BIT0 of bit parameter No.180 is set to 1) and 5 blocks of current program together. During the program execution, the blocks displayed are refreshed dynamically and the cursor is located in the block being executed.

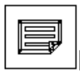


1.3.2 Program interface

1) PROGRAM CONTENT page

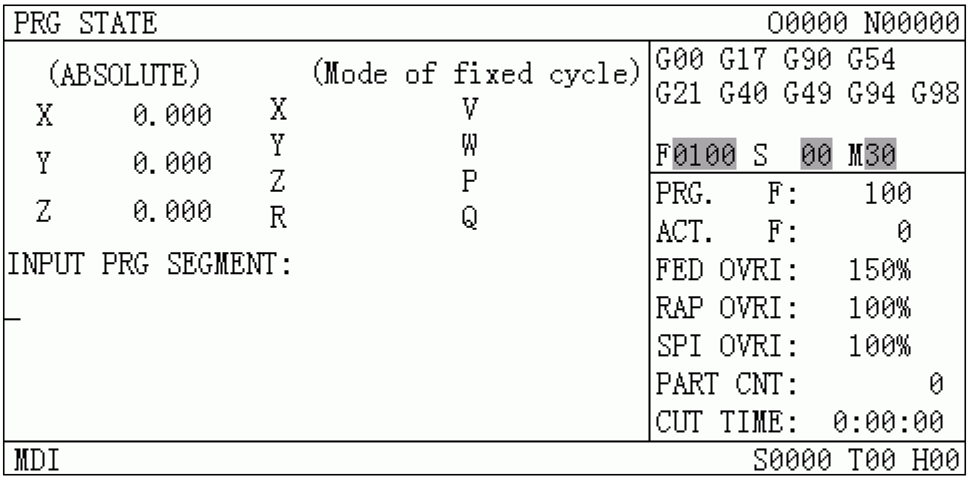
is a compound key.Press


key once to enter the program content interface，and

all blocks will be displayed by pressing  and keys in MDI mode.

2) PROGRAM STATE page

Press  key to enter program state interface in program content interface. Current G,M,S,T,F commands and related commands are displayed in program state interface and a single block （MDI）can be executed in this interface.



3) PROGRAM PREVIEW page

In program content interface，press  key to enter program preview page. In this page, all part programs are listed. To make it easier for user to select a program, the system displays 5 blocks before the program with cursor at the bottom of the page. User can press EOB directly to select a program and process automatically, or press DEL key to delete the program in this page. It displays the following contents :

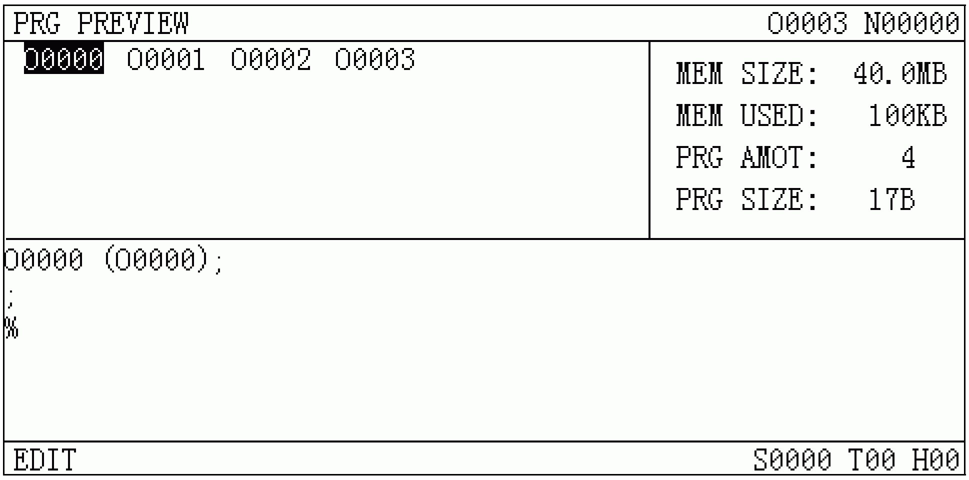
(a) Memory capacity: Display the maximum capacity of CNC memory unit.

(b) Used capacity：The space occupied by the saved programs

(c) Program NO.：Display the total number of programs in the CNC (including subprograms)

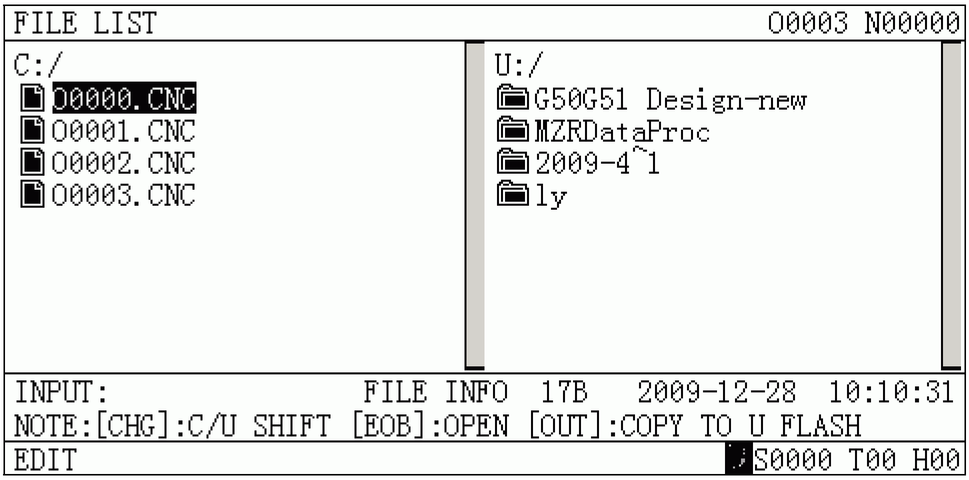
(d) Size of the program：The size of the program which the cursor is in, unit: byte (B)

(e) Program list：Display numbers of saved programs (arranged by name).



4) FILE LIST page

GSK980MDa supports USB interface, CNC→USB and USB→CNC mutual transmission operation are provided in this interface. In this page, it is easy to see the file list and file of CNC and USB (when USB is connected). At the same time, opening, duplication and deletion can be done here.



1.3.3 Tool offset, macro variable and tool life management interface

OFFSET is a compound key, press OFFSET key once in other page to enter the TOOL OFFSET page, press OFFSET key again to enter the MACRO interface.

1． OFFSET interface

There are 4 tool offset pages in this interface, and 32 offset numbers（No.001～No.032）available for user, which can be shown as the following figure by pressing  or  keys.

TOOL OFFSET O0003 N00000

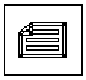
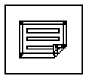
NO.	Geo(H)	Wear(H)	Geo(D)	Wear(D)
01	0.000	0.000	0.000	0.000
02	0.000	0.000	0.000	0.000
03	0.000	0.000	0.000	0.000
04	0.000	0.000	0.000	0.000
05	0.000	0.000	0.000	0.000
06	0.000	0.000	0.000	0.000
07	0.000	0.000	0.000	0.000
08	0.000	0.000	0.000	0.000

(RELATIVE)
X 0.000
Y 0.000
Z 0.000
(ABSOLUTE)
X 0.000
Y 0.000
Z 0.000

NO. 001

EDIT S0000 T00 H00

2． MACRRO interface

There are 25 pages in this interface, which can be shown by pressing  or  keys. In Macro page there are 600 （No.100～No.199 and No.500～No.999）macro variables which can be specified by macro command or set by keypad. Please refer to “macro, chapter 5, program” for related information.

MACRO O0003 N00000

NO.	DATA	NO.	DATA	NO.	DATA
_100	Null	108	Null	116	Null
101	Null	109	Null	117	Null
102	Null	110	Null	118	Null
103	Null	111	Null	119	Null
104	Null	112	Null	120	Null
105	Null	113	Null	121	Null
106	Null	114	Null	122	Null
107	Null	115	Null	123	Null

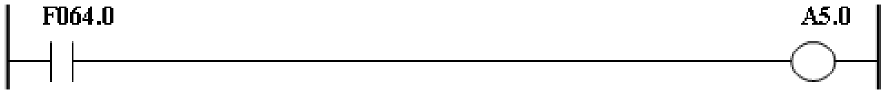
NO. 100

EDIT S0000 T00 H00

3. Tool life management

Note: The tool change signal TLCH：F064#0 should be added for PLC when using this function.

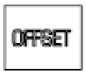
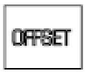
Ladder example:

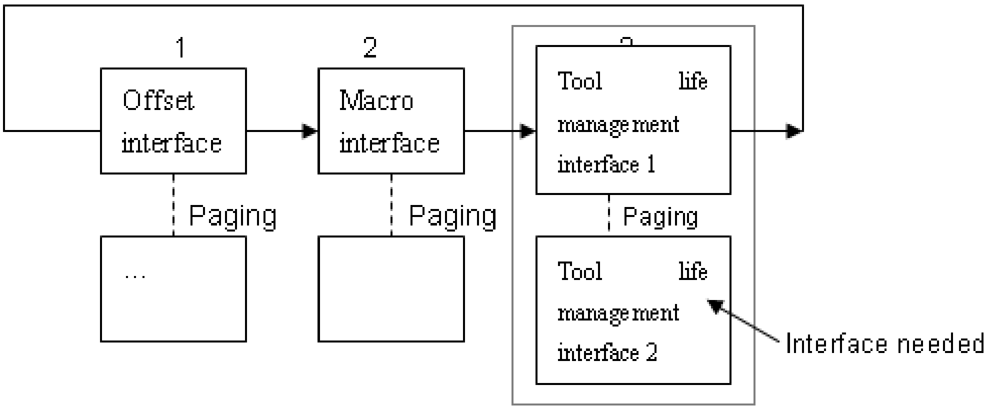


- **Using of tool life management function**

Parameter（No.002#0）is used as the symbol for tool life management function (0—unused，1—used); if the function is not used, the relevant tool life management page is not shown.

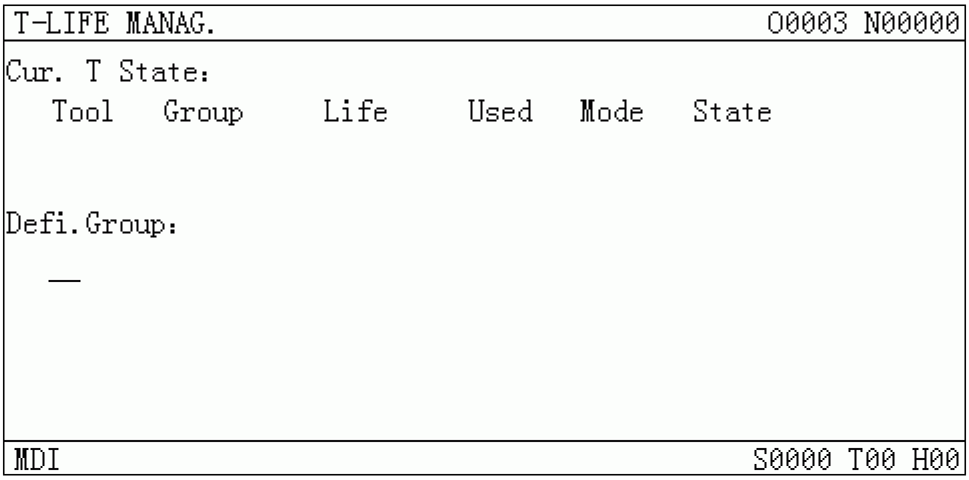
- **Tool life management display interface**

The tool life management is controlled by  key, which is displayed in the third sub-interface, and it is composed by 2 pages (paging by page keys). Interface is shown by pressing  key repeatedly



Tool life management display (the 1st page)

The 1st page for tool life management interface displays the life data of the current tool and the tool group list that has been defined. This page is mainly used for monitoring the tool life data by group units. The data monitoring of each tool in a group, group number setting and tool life management data are displayed in the following page.



ⅰ. Display explanation

<Current Tool State>: It displays the life data of the current tool which is being used.

Mode: It displays the counting unit of life data. (0: minute/1: times)

State: It displays the tool status. (0—Unused，1—Using，2—Over，3—Skip)

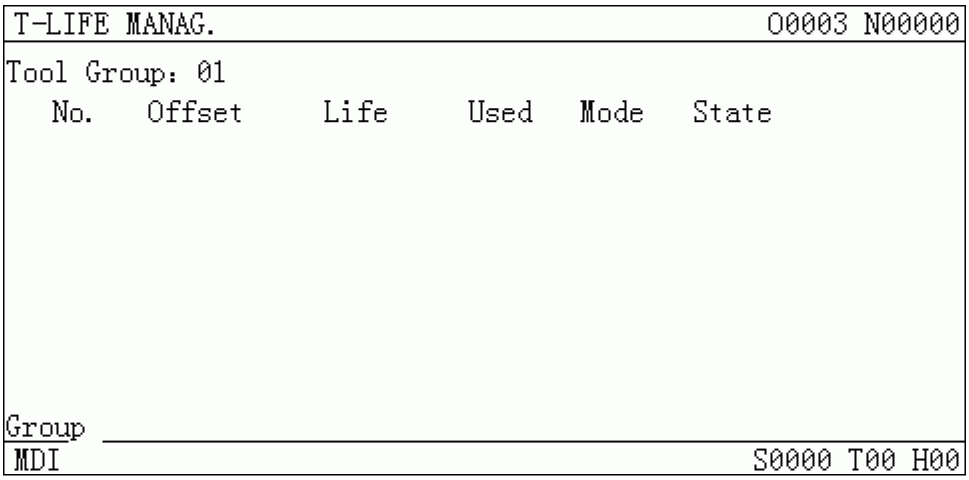
< Defined Group No. >： It only displays the group numbers which have been defined, and the undefined are not shown. The group number with the backlight means that all the tool life in that group has expired.

ⅱ. Deletion of all defined data

In this page, press CANCEL + G keys, it may delete all the data which have been defined (including group number, group tool numbers and life values, etc.)

Tool life management interface (the 2nd page)

The 2nd page is used to set and display the life data of a group which are displayed by order 1～8.



There are 3 display types for tool group selection:

i. Directly input the group number in the “Tool Group P”of the 2nd page, it displays the tool life

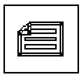
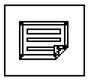
data. If the group does not exist, the number input will be taken as a new group number.The new group number: 05, and the 1st tool will be defined by system automatically:

ii. Move the cursor to select the group number in the "Defined Group No."of the 1st page, and it displays the group content as turning to the 2nd page.

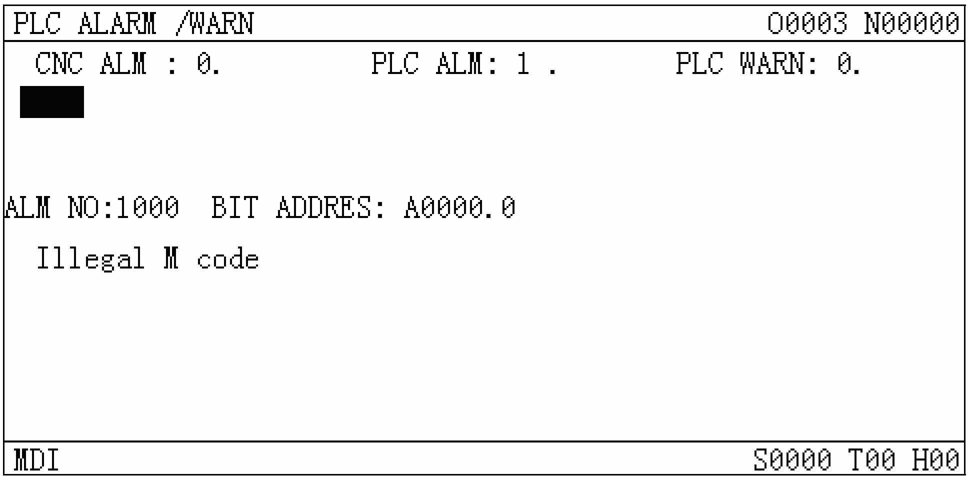
iii. As the current group number content is displayed in the 2nd page, it continues to display the following group number content by turning to the next page.

1.3.4 Alarm interface

Press 

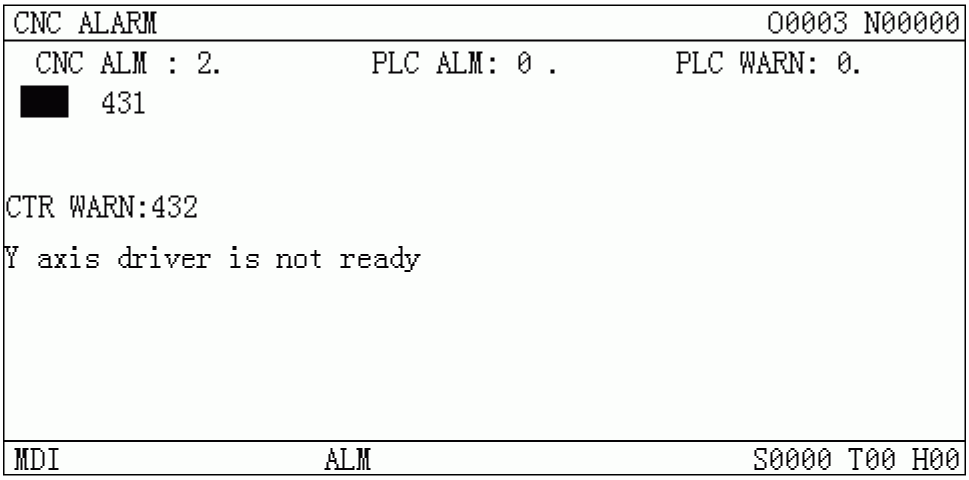
key to enter Alarm interface, there are CNC ALARM, PLC ALARM, ALARM LOG pages in this interface, which can be viewed by  or  key.

1) PLC ALARM: It displays the numbers of CNC alarm, PLC alarm and the current PLC alarm No., as well as PLC warning and warning No.. It may display 24 PLC alarm or warning No. together. The details for the respective alarm No. can be viewed by moving the cursor. The page is as follows：



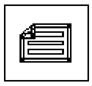
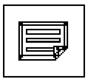
Page as the cursor locates at the alarm No.1000

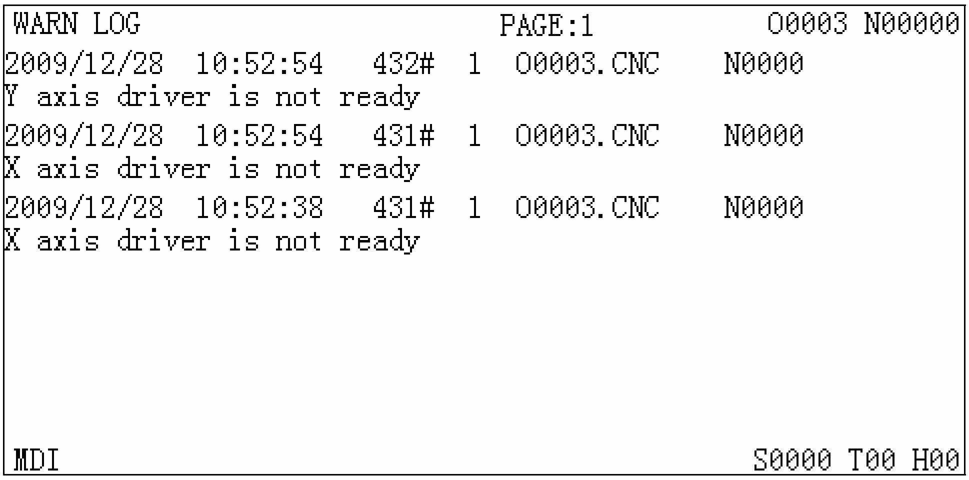
2) CNC ALARM: It displays the numbers of CNC alarm, PLC alarm and the current CNC alarm No.. It can display 24 CNC alarm No. together. The details for the respective alarm No. can be viewed by moving the cursor. The page is as follows:



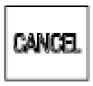
Page as the cursor locates at the alarm No.432

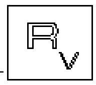
3) WARN LOG: Press 

key to enter Alarm interface, then press it again to enter the WARN LOG page, which records the latest alarm message including alarm date, alarm time, alarm No. and alarm content. 200 pieces warn log messages can be viewed by  or  key. See the following figure:

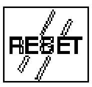


① **Sequence of warn log:** the latest alarm log message is shown on the forefront of the 1^{st} page, and the others queue in sequence. If the alarm log messages areover 200, the last one will be cleared.

② **Manual clearing of warn log:** under the 2 level authority, press 

+ 

key, it may clear all the warn log messages.

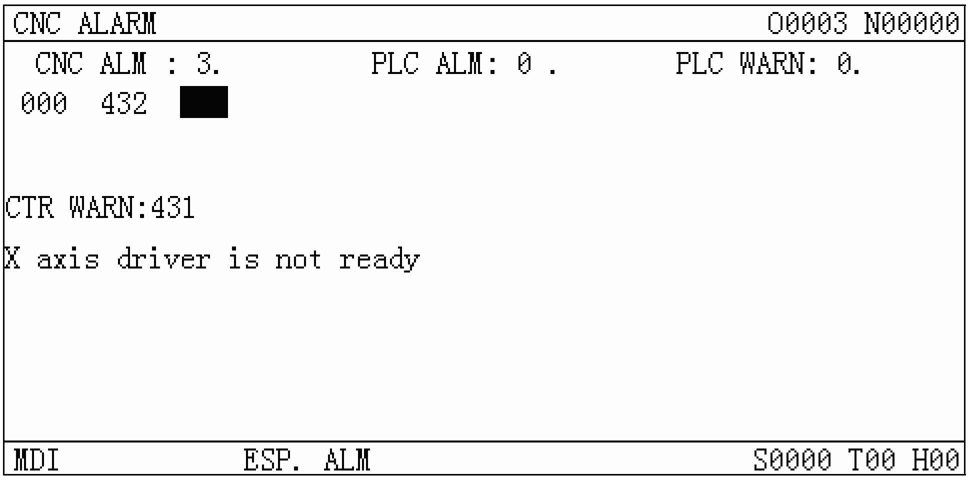
4）Alarm clearing: If multiple alarms are issued, only one alarm where the cursor locates could be cleared by pressing 

key each time (In alarm interface, it clears all alarms and warnings

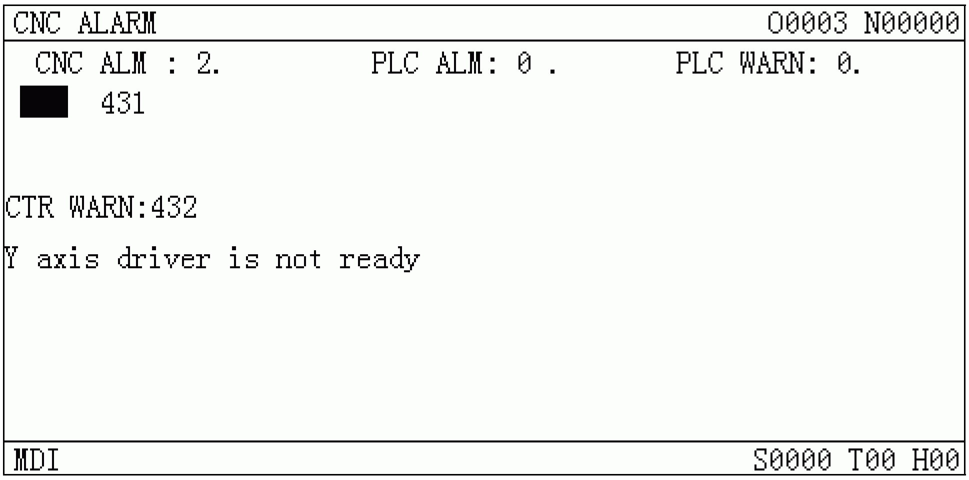
Volume II Operation

by pressing RESET and CANCEL keys).

5) The current alarm page is as florrows:



Current page



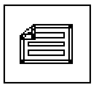
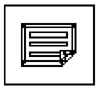
Page after pressing RESET key

6) Clearing PLC warning: If multiple warnings are issued, only one warning where the cursor locates could be cleared by pressing RESET or CANCEL key each time (In Alarm interface, it clears all alarms and warnings by pressing RESET and CANCEL keys).

1.3.5 Setting interface

SETTING is a compound key, press SETTING key in other page, it enters setting interface, press it again, it enters the G54~G59 interface, press it three times, it enters Graphic interface. Press SETTING key repeatedly, it switches among the above nentioned interfaces.

1.Setting interface

There are 3 pages in this interface, which can be viewed by  and  keys.

1）SWITCH SETTING: It is used for displaying the parameter, program, auto sequence No. on / off state.

PARM SWT: when it is turned ON, the parameters are allowed to be modified; it is turned OFF, the parameters are unallowed to be modified.

PROG SWT: when it is turned ON, the programs are allowed to be edited; it is turned OFF, the programs are unallowed to be edited.

AUTO SEG: when it is turned ON, the block No. is created automatically; it is turned OFF, the block No. is not created automatically, input manually if it is needed.

In this page, the state of on/off can be switched by ‘left / right’key or ‘U’and‘D’key on the MDI panel.

```
SWITCH SETTING                                  O0003 N00000

        ► PARM SWT:   OFF *ON
          PROG SWT:   OFF *ON
          AUTO SEG:  *OFF  ON

MDI                                           S0000 T00 H00
```

2）Data backup: In this page, the CNC data (bit parameter, data parameter, pitch parameter, tool offset) can be saved and restored.

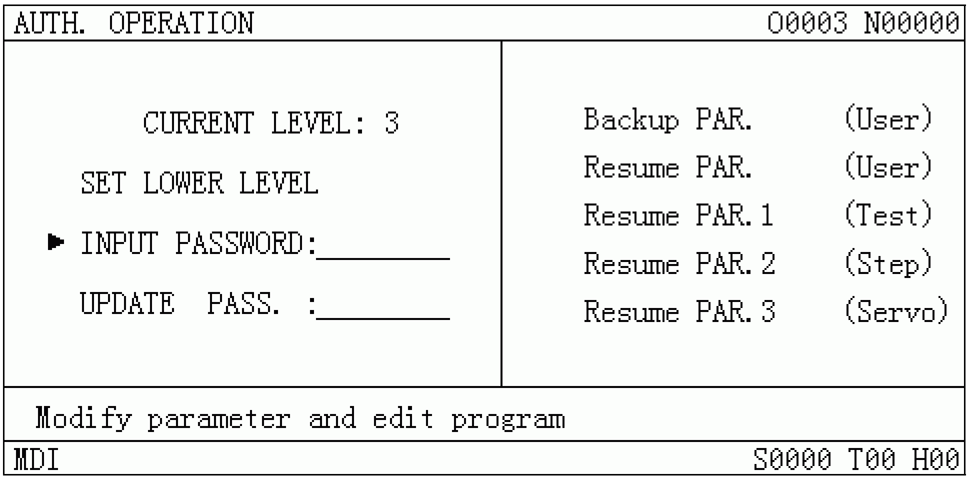
Data backup (user): For CNC data backup by user (save)

Recover backup data (user): For backup data recover by user (read)

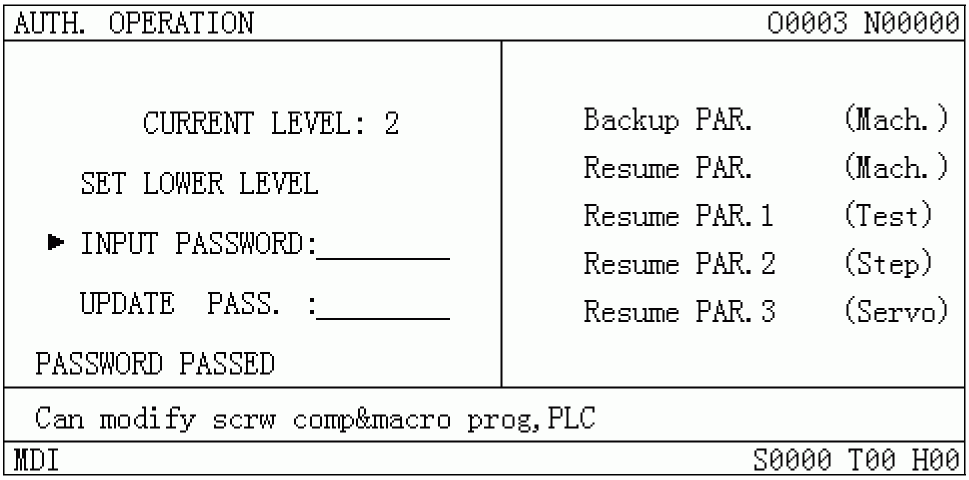
Recover standard parameter 1 (test): For reading original parameter data of CNC test by user

Recover standard parameter 2 (step): For reading original parameter data of suited step drive unit by user

Recover standard parameter 3 (servo): For reading original parameter data of suited servo drive unit by user.



User page of 3, 4, 5 level



User page of 2 level

3）Password setting：Display and set user operation level.

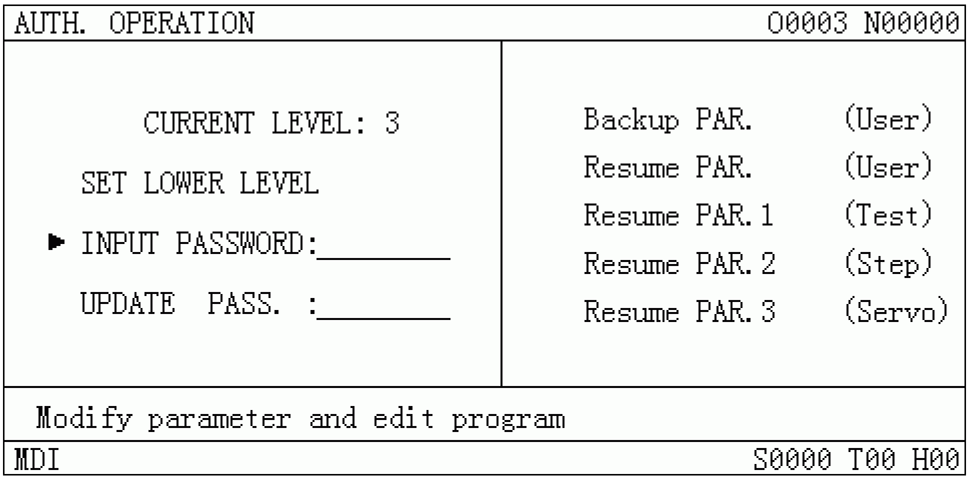
The password of GSK980MDa is composed of 4 levels, including machine builder (level 2), equipment management (level 3), technician (level 4) and machining operation (level 5).

Machine builder (level 2): It allows to modify CNC bit parameter, data parameter, screw- pitch parameter, tool offset parameter, edit part program (including macro program), edit and alter PLC ladder diagram, upload and download ladder diagram.

Equipment management (level 3): Initial password is 12345. The CNC bit parameter, data parameterm screw- pitch parameter, tool offset parameter, part program editing operations are allowed.

Technician (level 4): Initial password is 1234. Tool offset data (for tool setting), macro varibles, part program editing operations are allowed. However, CNC bit parameter, data parameter and pitch parameter editing operations are not allowed.

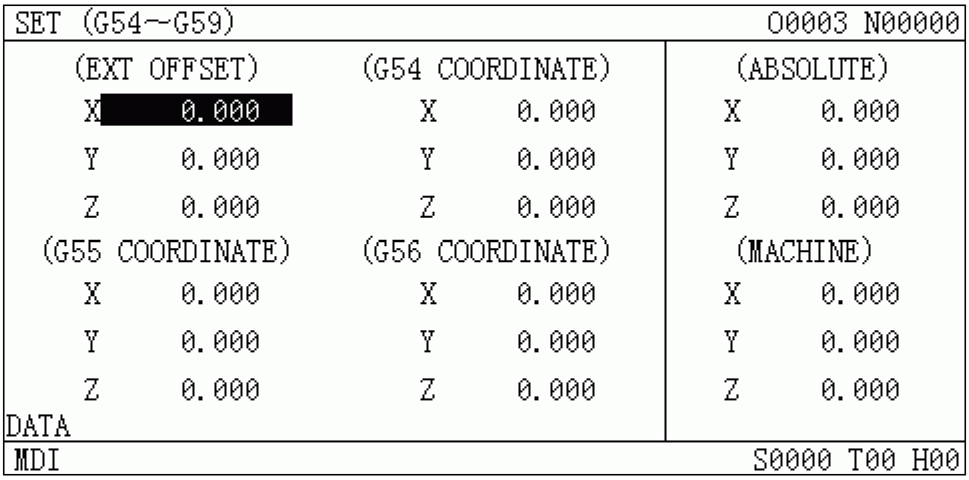
Machining operation (level 5): No password. Only the mschine panel operation is allowed. The alteration of tool offset data, CNC bit parameter, data parameter, pitch parameter, and the operations of part program selection, program editing are not allowed.

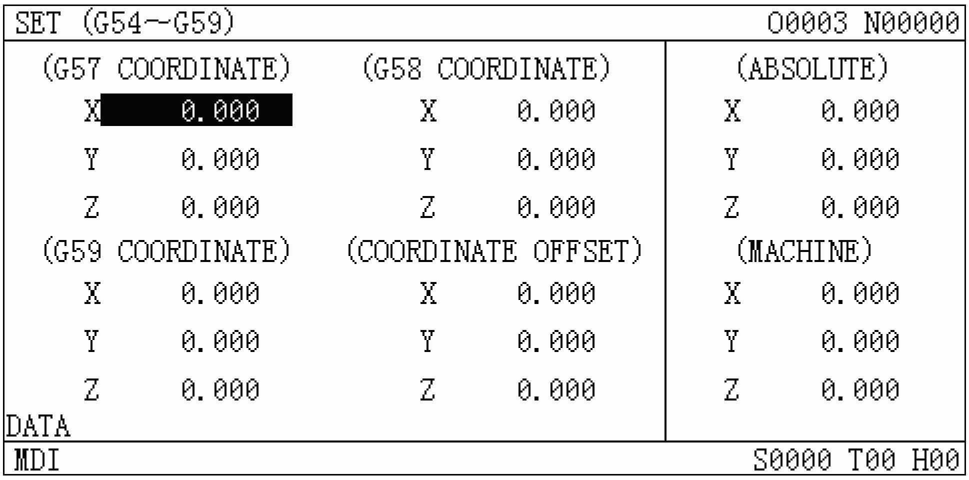


1.Setting page of G54～G59 Page location

Press 

key twice, this page is displayed.





The zero of the coordinate system: workpiece coordinate system zero offset, G54，G55，G56，G57，G58，G59.

- Moving of the cursor

  The cursor moves at the data of each coordinate system axis. And the data where the cursor

locates are highlighted.

The cursor supports up and down, left and right moving, and the corresponding data are backlighted.

By pressing Page key, the 1st group X axis data on the corresponding interface where the cursor locates is backlighted.

- Absolute data input

After “data+ DATA INPUT key” is keyed in by user, the data where the cursor locates is changed to the “data” input by user.

The validity judgement of user input data is the same as that of 980TD coordinate data input in MDI mode.

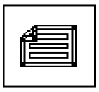
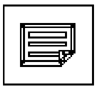
- Relative data input

After “data+ ALTER MACRO EDIT key” is keyed in by user, the original data where the cursor locates is changed by the sum of“data” newly input by user and original data.

- Auto measurement input

After “ X (or Z, Y)+ O + DATA INPUT key” is keyed in by user, the original data where the cursor locates is changed by the system current“X (or Z，Y) axis machine coordinate”.

3. Graphic interface

There are GRAPH SET, GRAPH TRACK pages in this interface, which can be viewed by  and  keys.

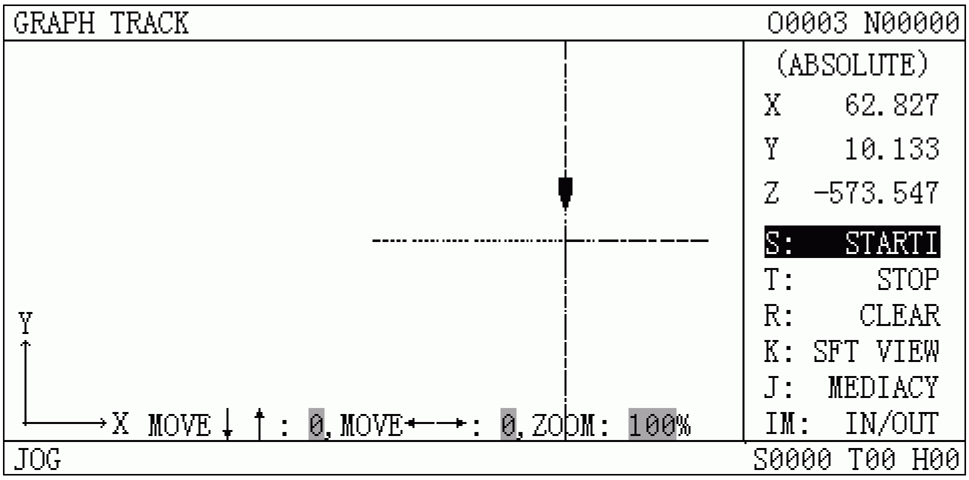
1）GRAPH SET page

In this page, the coordinate system, scaling and scope for graphic display can be selected.

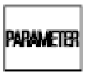
```
GRAPH SET                                        O0003 N00000
►COOR OPT=        0 0XY 1YX 2ZX 3XYZ 4YZ 5ZY 6XZ 7XZY)
 SCALE   =      100%
 CENTER  =     0.000 (X axis value)
 CENTER  =     0.000 (Y axis value)
 CENTER  =     0.000 (Z axis value)
 X MAX.  =   120.000
 Y MAX.  =   120.000
 Z MAX.  =   120.000
 X MIN.  =  -120.000
 Y MIN.  =  -120.000
 Z MIN.  =  -120.000
MDI                                           S0000 T00 H00
```

2）GRAPH TRACK page

In this page, it displays the path within the parameters range (refer to absolute coordinate) of GRAPH SET page.

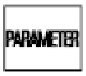


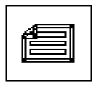
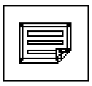
1.3.6 BIT PARAMETER, DATA PARAMETER, PITCH COMP interface

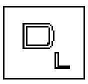


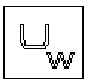
is a compound key, it enters BIT PARAMETER, DATA PARAMETER and PITCH COMP interfaces by pressing this key repeatedly.

1. BIT PARAMETER interface

Press 

key, it enters BIT PARAMETER interface, there are 48 bit parameters which are displayed by 2 pages in this interface, and they can be viewed or modified by pressing  or  key to enter the corresponding page. It is as follows:

As is shown in this page, there are 2 parameter rows at the bottom of the page, the 1st row shows the meaning of a bit of a parameter where the cursor locates, the bit to be displayed can be positioned by pressing 

or 

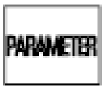
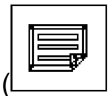
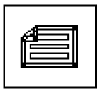
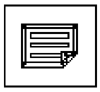
key. The 2nd row shows the abbreviation of all the bits of a parameter where the cursor locates.

BIT PARAMETER O0003 N00000

NO.	DATA	NO.	DATA	NO.	DATA
001	00000000	_009	00000000	017	00101000
002	00000011	010	00011111	018	00000000
003	00000000	011	00000000	019	10000000
004	01000000	012	00010011	020	00000000
005	00010001	013	10000011	021	00000000
006	00000000	014	00011111	022	00000000
007	00000000	015	10000000	023	00000000
008	00011111	016	00000000	024	00000000

*** *** *** ALM5 ALM4 ALMZ ALMY ALMX
bit7:1/0:Unused
NO. 009
JOG S0000 T00 H00

2. DATA PARAMETER interface

Press  key repeatedly ( key if in BIT PARAMETER interface), it enters DATA PARAMETER interface, there are 110 data parameters which are displayed by 7 pages in this interface, and they can be viewed or modified by pressing  or  key to enter the corresponding page. It is as follows:

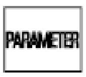
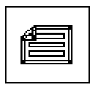
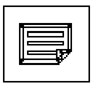
As is shown in this page, there is a cue line at the page bottom, it displays the meaning of the parameter where the cursor locates.

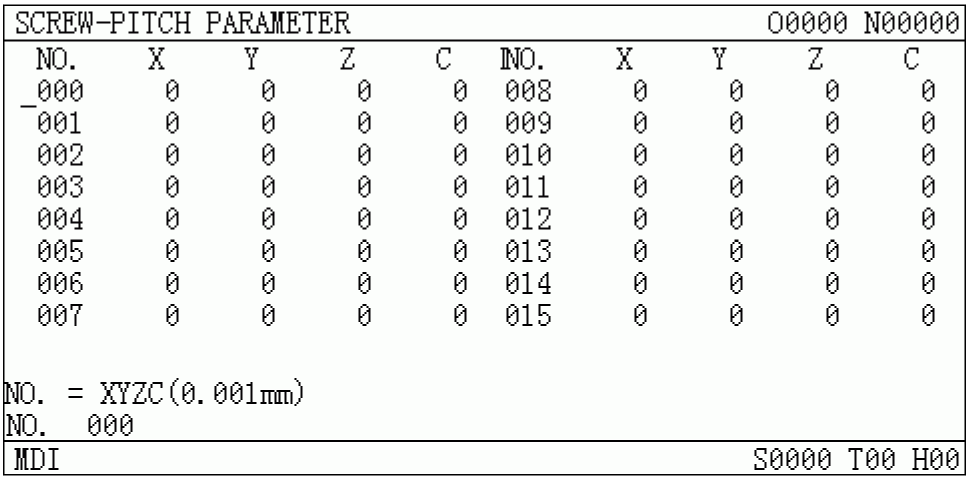
DATA PARAMETER O0003 N00000

NO.	DATA	NO.	DATA	NO.	DATA
_049	1	057	1	065	100
050	1	058	1	066	100
051	1	059	7600	067	100
052	1	060	7600	068	100
053	1	061	7600	069	400
054	1	062	7600	070	8000
055	1	063	7600	071	50
056	1	064	100	072	100

Command multiplier for X axis.
NO. 049
JOG S0000 T00 H00

- **PITCH COMP interface**

Press  key repeatedly, it enters PITCH COMP interface, there are 256 pitch parameters which are displayed by 16 pages in this interface, and they can be viewed by pressing  or  key.



SCREW-PITCH PARAMETER O0000 N00000

NO.	X	Y	Z	C	NO.	X	Y	Z	C
000	0	0	0	0	008	0	0	0	0
001	0	0	0	0	009	0	0	0	0
002	0	0	0	0	010	0	0	0	0
003	0	0	0	0	011	0	0	0	0
004	0	0	0	0	012	0	0	0	0
005	0	0	0	0	013	0	0	0	0
006	0	0	0	0	014	0	0	0	0
007	0	0	0	0	015	0	0	0	0

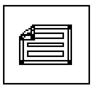
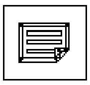
NO. = XYZC(0.001mm)
NO. 000
MDI S0000 T00 H00

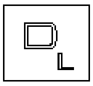
1.3.7 CNC DIAGNOSIS, PLC STATE, PLC VALUE, machine soft panel, VERSION MESSAGE interface

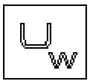
is a compound key, it enters CNC DIAGNOSIS, PLC STATE, PLC VALUE, machine soft panel, VERSION MESSAGE interfaces by pressing this key repeatedly.

1、CNC DIAGNOSIS interface CNC

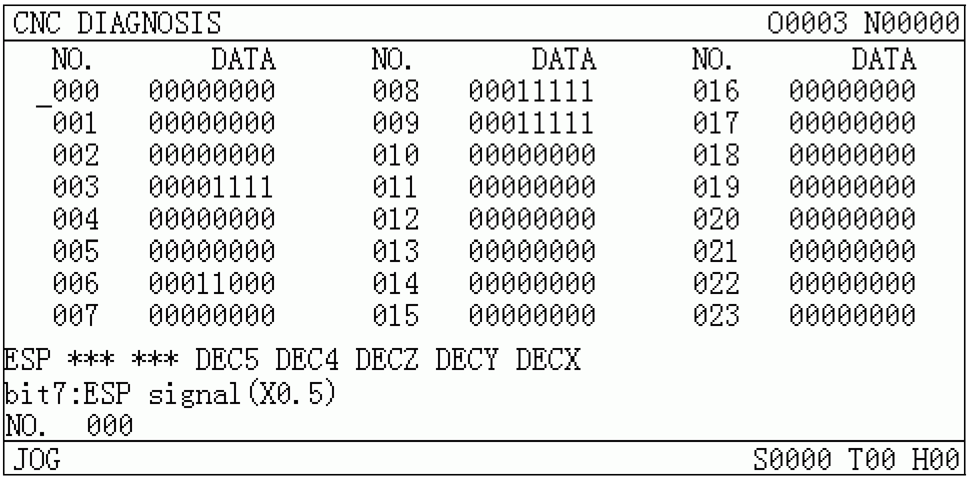
The input/output signal state between CNC and machine, the transmission signal state between CNC and PLC, PLC internal data and CNC internal state can all be displayed via diagnosis. Press 

key it enters CNC DIAGNOSIS interface, the keypad diagnosis, state diagnosis and miscellaneous function parameters etc. can be shown in this interface, which can be viewed by pressing  or  key.

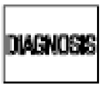
In CNC DIAGNOSIS page, there are 2 diagnosis No. rows at the page bottom, the 1st row shows the meaning of a diagnosis No. bit where the cursor locates, the bit to be displayed can be positioned by pressing 

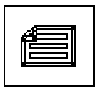
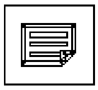
or 

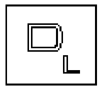
key. The 2nd row shows the abbreviation of all the diaosgnis No. bits where the cursor locates.

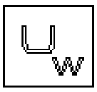


2. PLC STATE interface

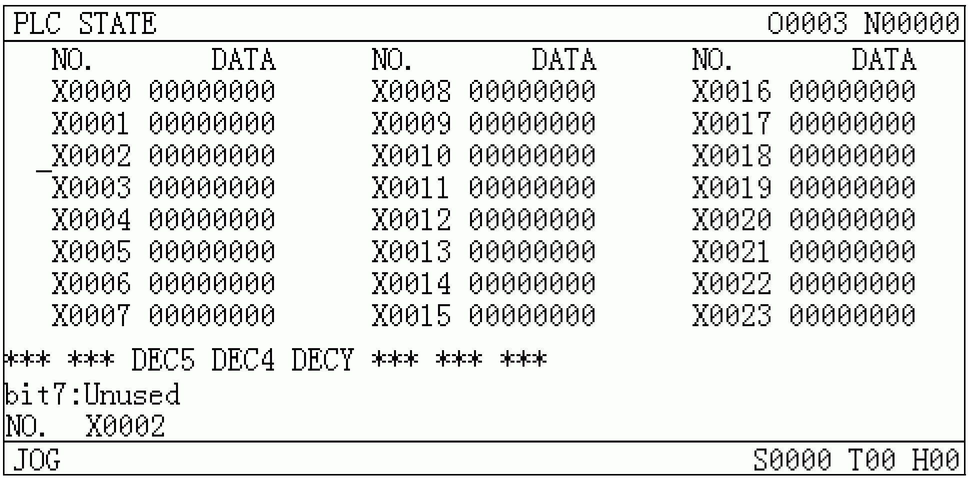
In the page of this interface, it orderly displays the state of address X0000~X0029, Y0000~Y0019, F0000~F0255, G0000~G0255, A0000~A0024, K0000~K0039, R0000~R0999 etc.. And it enters PLC STATE interface by pressing 

key repeatedly. The signal state of PLC addresses can be viewed by pressing  or  key.

In PLC STATE page, there are 2 rows at the page bottom; the 1st row shows the meaning of a bit of an address where the cursor locates, the bit to be displayed can be positioned by pressing 

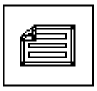
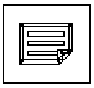
or 

key. The 2nd row shows the abbreviation of all the bits of an address where the cursor locates.



3. PLC VALUE interface

In the page of this interface, it orderly displays the values in the registers of T0000～T0099,D0000～D0999,C0000～C0099,DT000～DT099,DC000～DC099 etc.. By pressing 

key repeatedly it enters PLC VALUE interface. The data values of PLC can be viewed by pressing  or  key.

In this PLC VALUE page, there is a cue line at the page bottom, it displays the meaning of the parameter where the cursor locates. As is shown in the following figure:

PLC DATA O0003 N00000

NO.	DATA	NO.	DATA	NO.	DATA
DT000	0	DT008	0	DT016	0
DT001	0	DT009	0	DT017	0
DT002	0	DT010	0	DT018	0
DT003	0	DT011	0	DT019	100
DT004	0	DT012	0	DT020	500
DT005	0	DT013	0	DT021	500
DT006	0	DT014	0	DT022	100
DT007	0	DT015	0	DT023	500

Reserved
NO. DT000
JOG S0000 T00 H00

4. VERSION MESSAGE interface

It enters VERSION MESSAGE interface by pressing 

key repeatedly. The software, hardware, and PLC version message can be shown in this interface. The figure is as follows:

VERSION MESSAGE O0000 N00000

PRODUCT TYPE : GSK980MDa
SOFTWARE VER. : V2.00-manu
HARDWARE VER. : 3.01.002--08.07.21
SYSTEM ID: 0

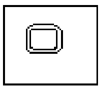
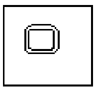
LADDER DESIGN: GSK
LADDER VER. : 09.01.15-839C
LADDER VERIFY: 839C
LADDER NOTE : GSK980MDa

MDI S0000 T00 H00

1.4 List of general operations

Item	Function	Operation key	Operation mode	Display page	Password level	Program on/off	Parameter switch
Clearing	Relative coordinate of X axis clearing	X . CANCEL		Relative coordinate			
	Relative coordinate of Y axis clearing	Y & . CANCEL		Relative coordinate			
	Relative coordinate of z axis clearing	Z ! . CANCEL		Relative coordinate			
	Part No. clearing	CANCEL + N		Relative coordinate or absolute coordinate			
	Cutting time clearing	CANCEL + T =					
	Tool radius offset D clearing	0. DATA INPUT		Tool offset	Level 2,3,4		
	Tool length offset H clearing	0. DATA INPUT		Tool offset	Level 2,3,4		
Data input	Bit parameter	Parameter. DATA INPUT	MDI mode	Bit parameter	Level 2,3		On
	Data parameter	Parameter. DATA INPUT	MDI mode	Bit parameter	Level 2,3		On

Volume II Operation

Item	Function	Operation key	Operation mode	Display page	Password level	Program on/off	Parameter switch
	Input pitch parameter of X axis	X. Compensation value. DATA INPUT	MDI mode	Pitch parameter	Level 2		On
	Input pitch parameter of Y axis	Y &. Compensation value. DATA INPUT	MDI mode	Pitch parameter	Level 2		On
	Input pitch parameter of Z axis	Z !. Compensation value. DATA INPUT	MDI mode	Pitch compensation parameter	Level 2		On
	Macro varibles	Macro varibles. DATA INPUT		Macro varibles	Level 2,3,4		
	Input tool radius offst D	Data value. DATA INPUT		Tool offst	Level 2,3,4		
	Input tool length offset H	Data value. DATA INPUT		Tool offset	Level 2,3,4		
Search	Search down from where the cursor locates	Character. ⇩	Edit mode	Program content	Level 2,3,4	On	
	Search up from where the cursor locates	Character. ⇧	Edit mode	Program content	Level 2,3,4	On	
	Search down from current program	. ⇩	Edit mode or auto mode	Program content, program list or program state	Level 2,3,4		
	Search up from current program	. ⇧			Level 2,3,4		

Item	Function	Operation key	Operation mode	Display page	Password level	Program on/off	Parameter switch
	Search defined program	O. program name. ⇩			2 级. 3 级. 4 级 Level 2,3,4		
	Search for bit parameter, data parameter or pitch parameter	P Q. Parameter no.. DATA INPUT		Corresponding page of the data			
	PLC state, PLC data searching	P Q. address No. . DATA INPUT		PLC state, PLC data			
Deletion	Delete the character where the cursor is in	DELETE	Edit mode	Program content	Level 2,3,4	On	
		CANCEL	Edit mode	Program content	Level 2,3,4	On	
	Single block deletion	Move the cursor to the head of the line. DELETE	Edit mode	Program content	Level 2,3,4	On	
	Multi-block deletion	CHANGE. N. order number. DELETE	Edit mode	Program content	Level 2,3,4	On	
	Segment deletion	CHANGE. character. DELETE	Edit mode	Program content	Level 2,3,4	On	

Item	Function	Operation key	Operation mode	Display page	Password level	Program on/off	Parameter switch
	Delete one program	. program name. DELETE	Edit mode	Program content	Level 2,3,4	On	
	Delete all programs	. 999. DELETE	Edit mode	Program content	Level 2,3,4	On	
Change name	Change program name	. program name. ALTER MACRO EDIT	Edit mode	Program content	Level 2,3,4	On	
Duplication	Duplicate program	. program name. CHANGE	Edit mode	Program content	Level 2,3,4	On	
CNC →CNC （send）	Tool offset	DATA OUTPUT	Edit mode	Tool offset	Level 2,3		On
	Bit parameter	DATA OUTPUT	Edit mode	Bit parameter	Level 2,3		On
	Data parameter	DATA OUTPUT	Edit mode	Data parametr	Level 2,3		On
	Pitch parameter	DATA OUTPUT	Edit mode	Pitch parameter	Level 2		On
	Send a part program	, program name, DATA OUTPUT	Edit mode	Program content	Level 2,3,4	On	

Item	Function	Operation key	Operation mode	Display page	Password level	Program on/off	Parameter switch
	Send all part programs	. 999. DATA OUTPUT	Edit mode	Program content	Level 2,3,4	On	
CNC →CNC（receive）	Tool offset		Edit mode		Level 2,3,4		On
	Bit parameter		Edit mode		Level 2,3		On
	Data parameter		Edit mode		Level 2,3		On
	Pitch parameter		Edit mode		Level 2		On
	Part program		Edit mode		Level 2,3,4	On	
CNC →PC (upload)	Tool offset	DATA OUTPUT	Edit mode	Tool offset	Level 2,3,4		On
	Bit parameter	DATA OUTPUT	Edit mode	State parameter	Level 2,3,4		On
	Bit parameter	DATA OUTPUT	Edit mode	Data parameter	Level 2,3		On
	Pitch parameter	DATA OUTPUT	Edit mode	Pitch compensation parameter	Level 2		On
	Send a program	, program name, DATA OUTPUT	Edit mode	Program content	Level 2,3,4	On	
	Send all programs	. 999. DATA OUTPUT	Edit mode		Level 2,3,4	On	

Volume II Operation

Item	Function	Operation key	Operation mode	Display page	Password level	Program on/off	Parameter switch
PC→CNC (download)	Tool offset		Edit mode		Level 2,3,4		On
	Bit parameter		Edit mode		Level 2,3		On
	Data parameter		Edit mode		Level 2,3		On
	Pitch parameter		Edit mode		Level 2		On
	Part program		Edit mode		Level 2,3,4	On	
Switch setting	Turn on parameter switch	D L		Switch setting	Level 2,3		
	Turn on program switch	D L		Switch setting	Level 2,3,4		
	Turn on auto sequence No.	D L		Switch setting			
	Turn off parameter switch	U W		Switch setting	Level 2,3		
	Turn off program switch	U W		Switch setting	Level 2,3,4		
	Turn off auto sequence No.	U W		Switch setting			

Explanations: ". " in the column "operation" indicates operate two keys successively, "+" indicates operate two keys simultaneously.

Example: **U W . CANCEL indicates that press U W key first，and then press CANCEL key; CANCEL + N indicates that press two keys simultaneously.**

CHAPTER 2 POWER ON OR OFF AND PROTECTION

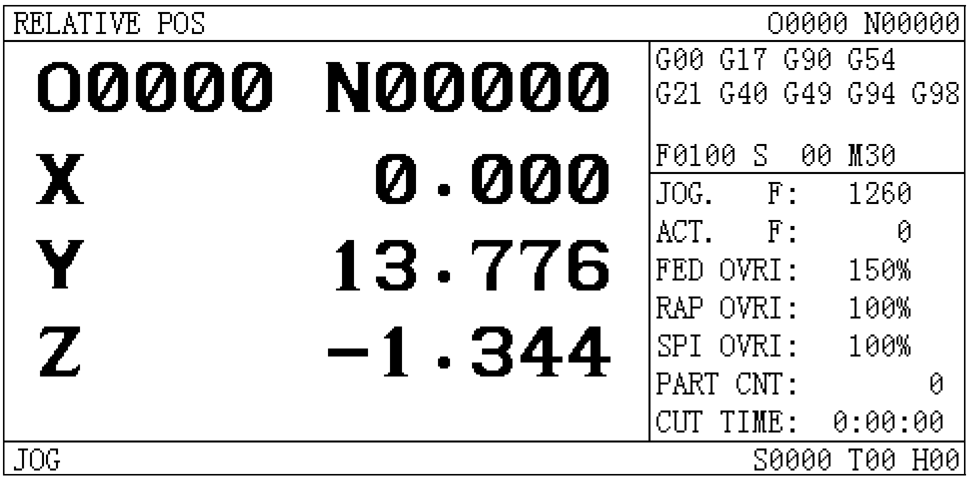
2.1 System Power On

Before this GSK980MDa is powered on, the following should be confirmed:

1. The machine is in a normal state.
2. The power voltage conforms to the requirement of the machine.
3. The connection is correct and secure.

The following page is displayed after GSK980MDa is powered on:

The current position (RELATIVE POS) page is displayed after system auto detection and initiation are finished.



2.2 System Power Off

Before power is off, ensure that:

1. The axes of the CNC are at halt;
2. Miscellaneous functions (spindle, pump etc.) are off;
3. Cut off CNC power prior to machine power cutting off.

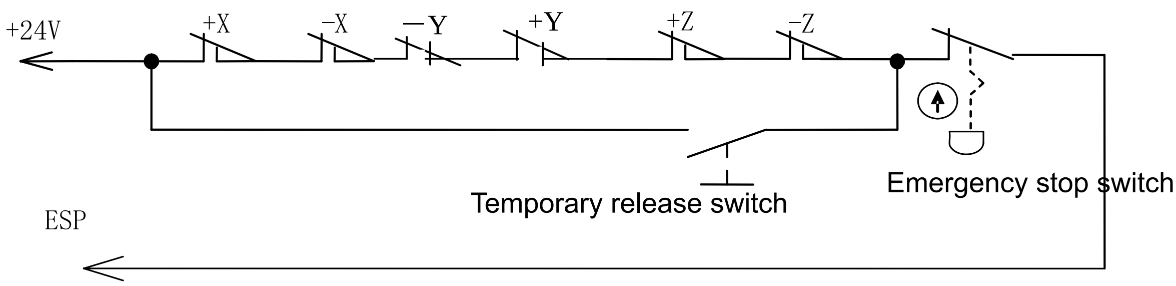
Note: Please see the machine builder's manual for the machine power cut-off operation.

2.3 Overtravel Protection

Overtravel protection should be employed to prevent the damage to the machine due to the overtravel of the axes.

2.3.1 Hardware overtravel protection

The stroke switches are fixed at the positive and negative maximum travel of the machine axes X, Y, Z, 4th, 5th respectively，they are connected by the following figure. And the “MESP”of bit parameter No.017 must be setted to 0. If the overtravel occurs, the stroke switch acts to make the machine stop, and the emergency alarm issues.



When the hardware overtravel occurs, there will be an “emergency stop”alarm. The steps to eliminate this alarm is press the OVERTRAVEL button to reversely move the table to detach the stroke switch (for positive overtravel, move negatively; vice versa).

2.3.2 Software overtravel protection

When the “MOT” of bit parameter No.17 is set to 0, the software limit is valid.

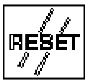
The software travel stroke is set by data parameter NO.135~ NO.144, they refer to machine coordinate. No.135~No.139 are for axes (X, Y, Z, 4th, 5th) positive max.overtravel, №140～№144 are for negative max.overtravel.

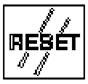
If the machine position (coordinate) exceeds the setting range, overtravel alarm will occur. The steps to eliminate this alarm is press RESET key to clear the alarm, then moves reversely (for positive overtravel, move out negatively; vice versa)

2.4 Emergency Operation

During the machining, some unexpected incidents may occur because of the user programming, operation and product fault.So this GSK980MDa should stopped immediately for these incidents. This section mainly describes the resolutions that this GSK980MDa are capable of under the emergency situation. Please see the relative explanation for these resolutions under the emergency by machine builder.

2.4.1 Reset

Press  key to reset this GSK980MDa system if there are abnormal outputs and axis actions in it:

1. All axes movement stops;
2. M, S function output is invalid (PLC ladder defines whether automatically cut off signals such as spindle CCW/CW, lubrication, cooling by pressing  key);
3. Auto run ends, modal function and state held on.

2.4.2 Emergency stop

During machine running, if the emergency button is pressed under the dangerous or emergent situation, the CNC system enters into emergency status and the machine movement is stopped immediately. If the emergency button is released, the emergency alarm is cancelled and the CNC resets. Its circuit wiring is shown in section 2.2.1 of this chapter.

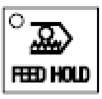
Note 1 Ensure the fault is eliminated before the emergency alarm is cancelled.

Note 2 pressing down the Emergency button prior to power on or off may alleviate the electric shock to the machine system.

Note 3 Reperform the machine zero return operation to ensure the correct position coordinate after the emergency alarm is cancelled (machine zero return operation is unallowed if there is no machine zero on the machine.).

Note 4 Only the MESP of the bit parameter No.017 is set to 0, is the external emergency stop valid.

2.4.3 Feed hold

Key can be pressed during the machine running to make the running pause. However, in thread cutting, cycle running, this function can not stop the running immediately.

2.4.4 Power off

Under the dangerous or emergency situations during the machine running, the machine power should be cut off immediately to avoid the accidents. However, it should be noted that there may be a big error between the CNC displayed coordinate and the actual position. So the tool setting operation should be performed again.

CHAPTER 3 MANUAL OPERATION

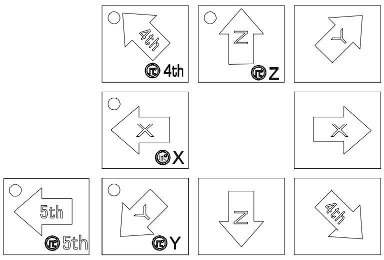
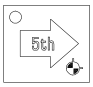
Press  key, it enters Manual mode. In this mode, the manual feed, spindle control, override adjustment operations can be performed.

> **Note!**
>
> The keys functions of this 980MDa machine panel are defined by Ladder Diagram; please refer to the respective materials by the machine builder for the function significance.
>
> Please note that the following function introduction is described based on the 980MDa standard PLC programs!

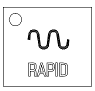
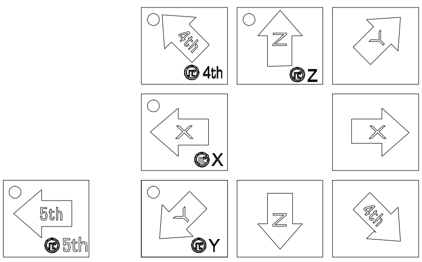
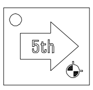
3.1 Coordinate axis moving

In Manual mode, the coordinate axis can be moved manually for feeding and rapid traverse.

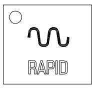
3.1.1 Manual feed

Press feed axis and axis direction key in the direction selection area  ,  the corresponding axis may be moved positively or negatively, and the axis stops moving if releasing these two keys; and the direction selection keys of X. Y. Z. 4th. 5th axes can be hold on at a time to make the 5 axes to move simultaneously.

3.1.2 Manual rapid traverse

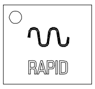
First press  key in the feed axis and direction selection area   till the rapid traverse indicator in the State area lights

up. The corresponding axis can be rapidly moved positively or negatively by pressing direction selection key, and the axis stops moving if releasing the key; and the direction selection keys of X. Y. Z. 4th. 5th axes can be hold on at a time to make the 5 axes to move simultaneously.

In Manual rapid mode, press 

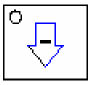
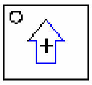
key to make the indicator go out, and the rapid traverse is invalid, it enters the Manual feed mode.

Note 1: Before machine zero return, the validity of manual rapid traverse is set by the "ISOT" of the bit parameter No.012.

Note 2: In Edit or MPG mode, 

key is invalid.

3.1.3 Manual feedrate override adjustment

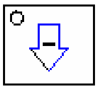
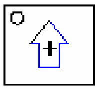
In Manual mode, the  or  key in 

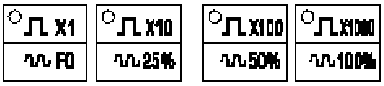
can be pressed to modify the Manual feedrate override, and the override has 16 levels. The relation of the feedrate override and the feedrate is as the following table:

Feedrate override (%)	Feedrate (mm/min)
0	0
10	2.0
20	3.2
30	5.0
40	7.9
50	12.6
60	20
70	32
80	50
90	79
100	126
110	200
120	320
130	500
140	790
150	1260

Note: There is about 2% fluctuating error for the data in the table.

3.1.4 Manual rapid override adjustment

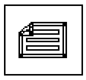
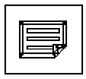
In the manual rapid traverse,  or  key in 

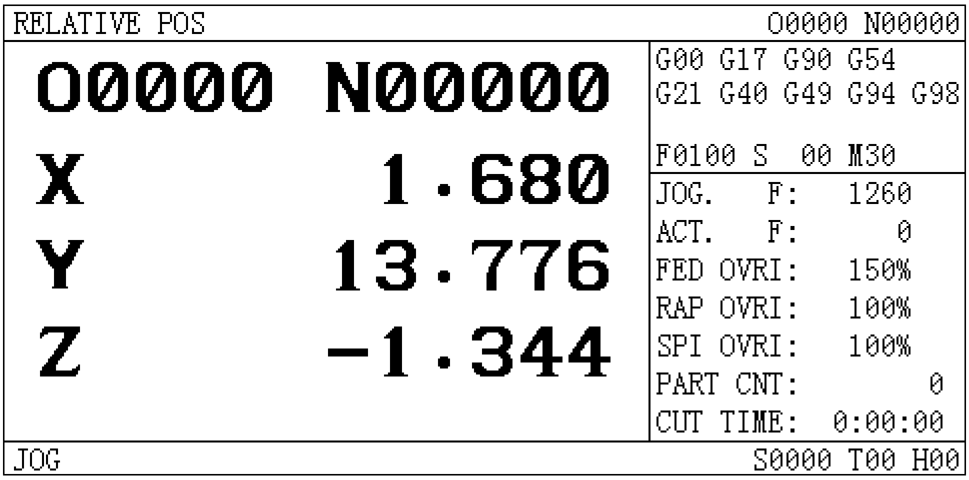
can be pressed (also by 

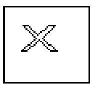
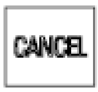
key with the respective override F0, 25%,50%， 100%)to modify the Manual rapid override, and there are 4 gears of F0, 25%, 50%， 100% for the override.(F0 is set by data parameter No.069)

3.1.5 Relative coordinate clearing

1）Press 

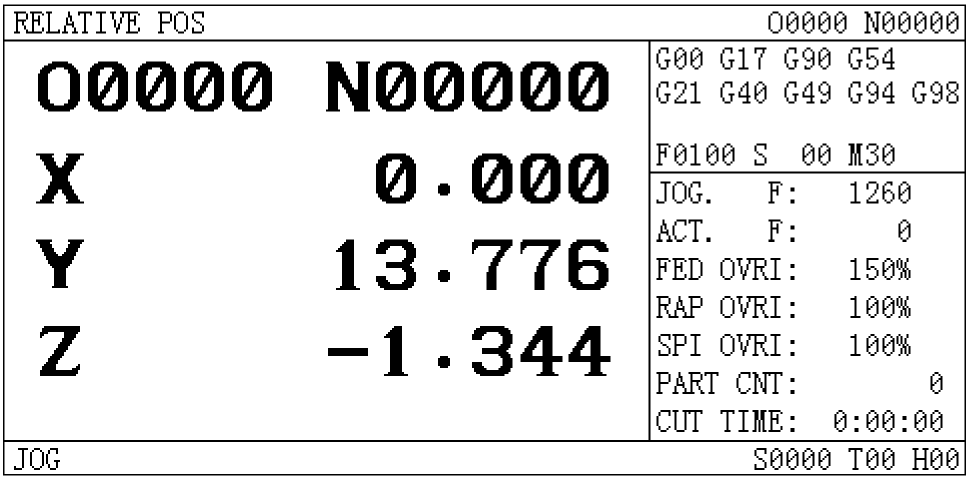
key to enter Position interface, then press  or  key to select the RELATIVE POS page;



2）Press  key to make the “X”in the page to blink,then press 

key;

Volume II Operation

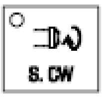


3）The clearing operations of other coordinates are the same as above.

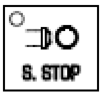
3.2 Other Manual operations

Note: The following operations are also valid in Machine zero, MPG/Step mode.

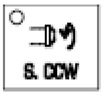
3.2.1 Spindle CCW, CW, stop control



： In Manual mode, the spindle rotates conterclockwise if pressing this key;；

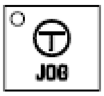


： In Manual mode, the spindle stops if pressing this key;



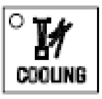
： In Manual mode, the spindle rotates clockwise if pressing this key;

3.2.2 Spindle Jog

Press and hold 

key, the spindle rotates conterclockwise, release it, the spindle stops.

3.2.3 Cooling control



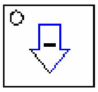
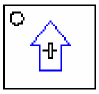
： In Manual mode, press this key, the coolant is switched on/off.。

3.2.4 Lubrication control

See details in Appendix for its function.

3.2.5 Spindle override adjustment

In Manual mode, if the spindle speed is controlled by analog voltage output, the spindle speed may be adjusted.

By pressing the  or  key in Spindle Override keys 

, the spindle speed can be changed by real-time adjusting of the spindle override that has 8 levels of 50％～120％.

CHAPTER 4 MPG/STEP OPERATION

In MPG/Step mode, the machine moves by a specified increment.

Note!

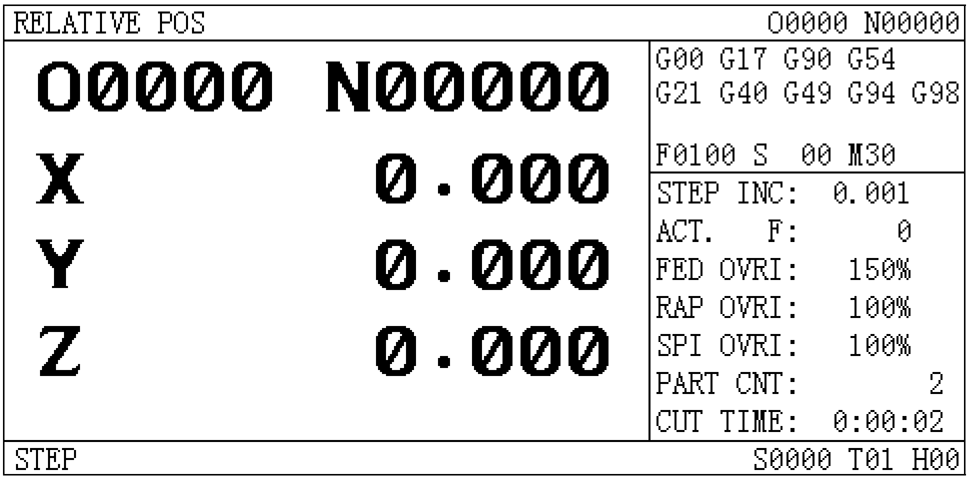
The keys functions of this 980MDa machine panel are defined by Ladder; please refer to the respective materials by the machine builder for the function significance.

Please note that the following function introduction is described based on the 980MDa standard PLC programs!

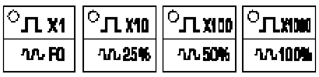
4.1 Step Feed

Set the BIT3 of the bit parameter No.001 to 0, and press 

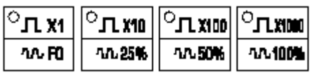
key to enter the Step mode, it displays as follows:



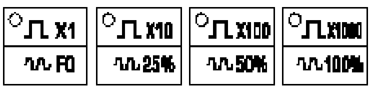
4.1.1 Increment selection

Press 

key to select the move increment, the increment will be shown in the page..

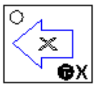
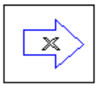
Note: In the EDIT or REF modes, 

keys are invalid. In the AUTO or MDI modes, rapid override will be changed by pressing the above-mentioned keys. In the MANUAL mode, press rapid move key 

and 

keys together, these keys are valid, otherwise, they are invalid.

4.1.2 Moving direction selection

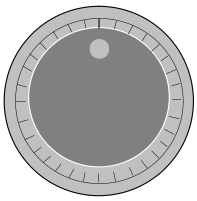
Press  or  key once, X axis can be moved negatively or positively by a step increment, other axises are the same.

4.2 MPG (Handwheel) Feed

Set the BIT3 of the bit parameter No.001 to 1, and press  key to enter the MPG mode, it displays as following:

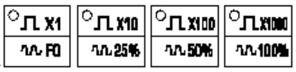
```
RELATIVE POS                                  O0000 N00000
 O0000  N00000            | G00 G17 G90 G54
                          | G21 G40 G49 G94 G98
                          |
 X            0.000       | F0100 S   00 M30
                          | HNDL INC:   0.001
 Y            0.000       | ACT.    F:        0
                          | FED OVRI:    150%
 Z            0.000       | RAP OVRI:    100%
                          | SPI OVRI:    100%
                          | PART CNT:         2
                          | CUT TIME:   0:00:02
HNDL                                       S0000 T01 H00
```

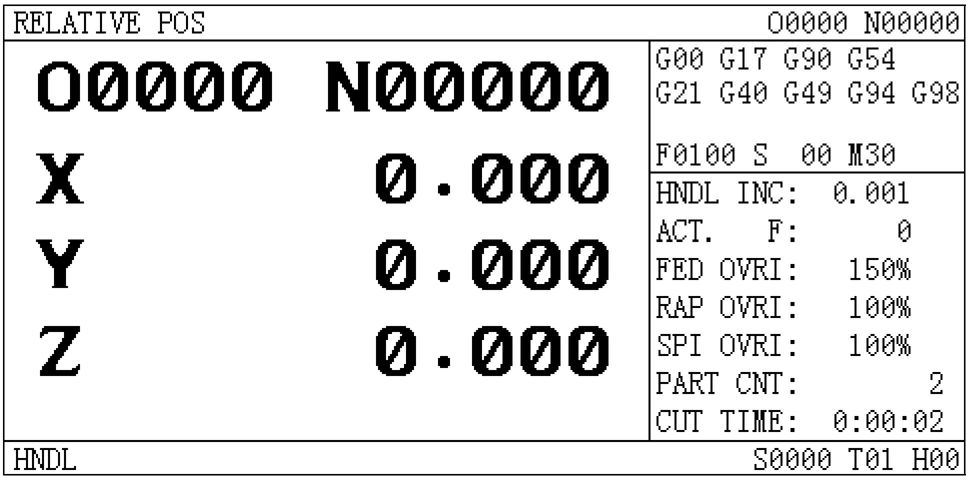
The handwheel figure is as follows：



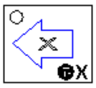
The handwheel figure

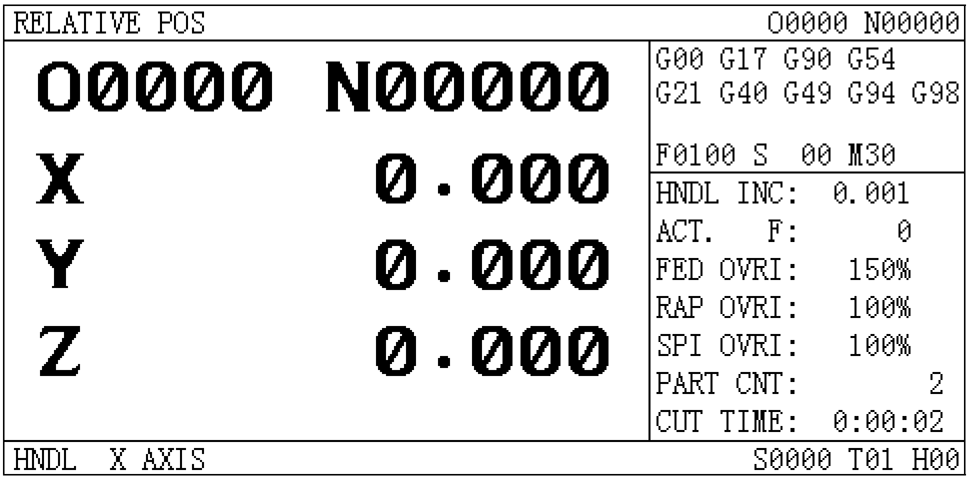
4.2.1 Increment selection

Press  key to select the move increment, the increment will be shown in the page：



4.2.2 Moving axis and direction selection

In MPG mode, press  key to select the corresponding axis. The page is as follows (Other axises are the same):



The handwheel feed direction is defined by its rotation direction. Generally, the handwheel CW is for positive feed, and CCW is for negative feed. In case of that handwheel CW is for negative feed, CCW for positive feed, it may exchange the A, B signals of the handwheel terminals,also you can modify the HNGX. HNGY. HNGZ. HNG4. HNG5 of the bit parameter №019.

4.2.3 Explanation items

1. The correspondence between the handwheel scale and the machine moving amount is as following table:

	Moving amount of each handwheel scale			
Handwheel increment	0.001	0.0100	0.100	1.000
Specified coordinate value	0.001mm	0.010mm	0.100mm	1.000mm

2. The rotation speed of the handwheel should be less than 5 r/s, if it is over that, the scale may be not coincide with the moving amount
3. The handwheel axis selection key is valid only in the MPG mode.

CHAPTER 5 MDI OPERATION

In MDI mode, the operations of parameter setting, words input and execution can be performed.

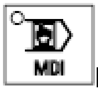
Note!

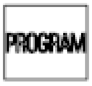
The keys functions of this 980MDa machine panel are defined by Ladder; please refer to the respective materials by the machine builder for the function significance.

Please note that the following function introduction is described based on the 980MDa standard PLC programs!

5.1 Code Words Input

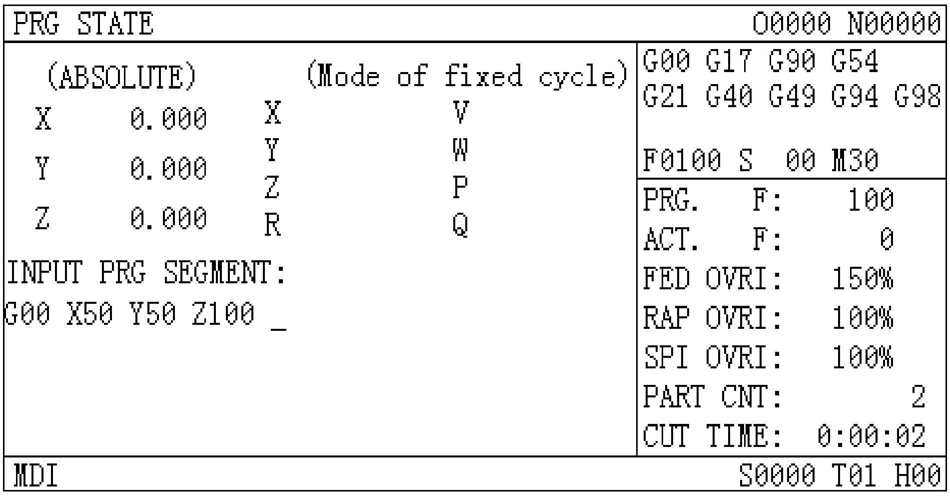
Select MDI mode to enter the PRG STATE page, to input an block “G00 X50 Z100”，the steps are as follows:

1. Press  key to enter MDI mode;

2. Press  key to enter PRG STATE page:

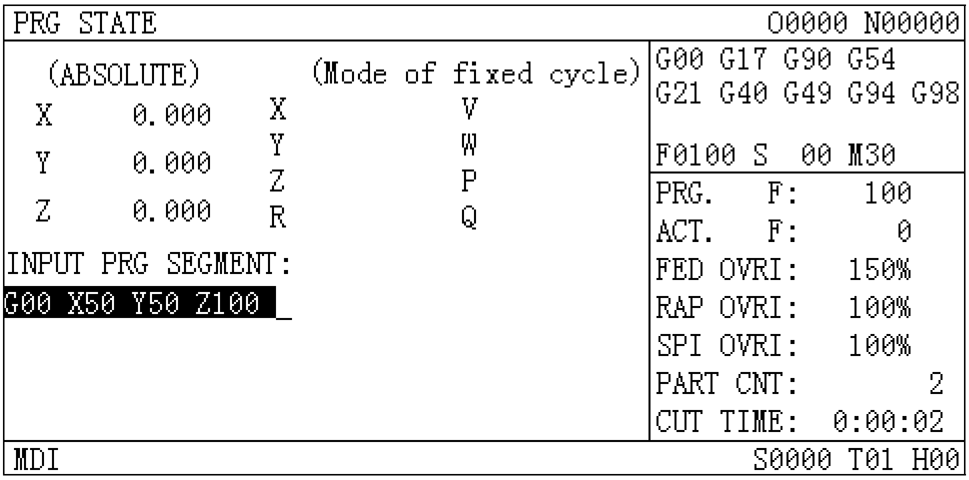

```
PRG STATE                                                O0000 N00000
 (ABSOLUTE)        (Mode of fixed cycle)  | G00 G17 G90 G54
 X      0.000   X          V              | G21 G40 G49 G94 G98
 Y      0.000   Y          W              |
                Z          P              | F0100 S  00 M30
 Z      0.000   R          Q              | PRG.    F:     100
                                          | ACT.    F:       0
INPUT PRG SEGMENT:                        | FED OVRI:   150%
_                                         | RAP OVRI:   100%
                                          | SPI OVRI:   100%
                                          | PART CNT:        2
                                          | CUT TIME:  0:00:02
MDI                                              S0000 T01 H00
```

3. Input G、0、0、X、5、0、Y&、5、0、Z;、1、0、0 by sequence, the page is as follows:



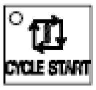
4. Press 

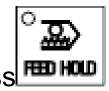
, the page is as follows:

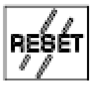


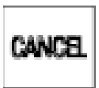
5.2 Code Words Execution

After the words are input, and press 

, the background color of program segment becomes white, these MDI words are executed after the 

key is pressed. During the execution,Press 

, 

and Emergency Stop button may be pressed to terminate the MDI words execution.If 

key is pressed,the background color of program segment will becomes black,then words can be input again.

Note:　The subprogram call command (M98 P　　; etc.) is invalid in MDI mode.

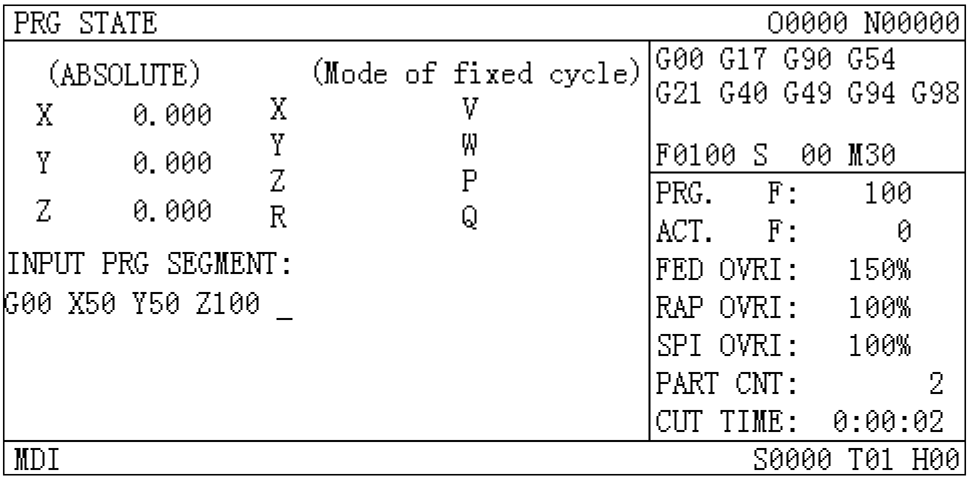
5.3 Parameter Setting

In MDI mode, the parameter value can be modified after entering the parameter interface. See details in Chapter 9 of this part.

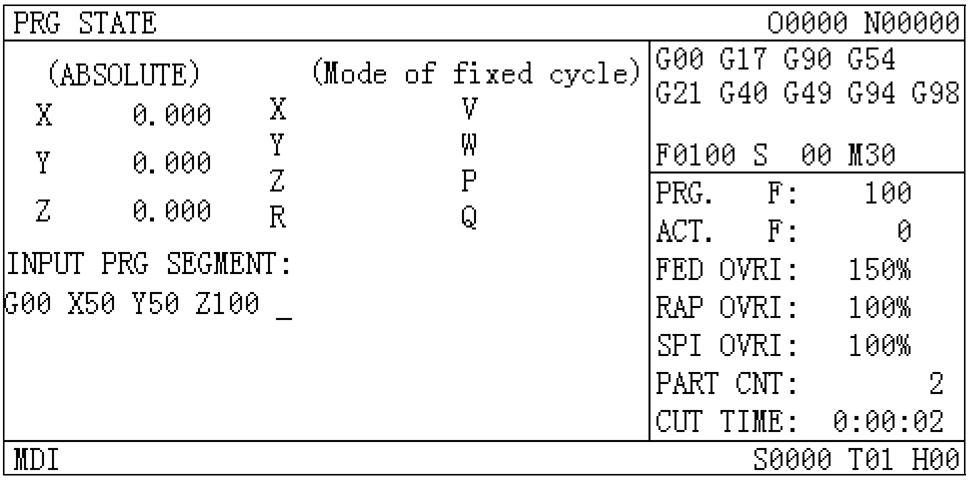
5.4 Data Modification

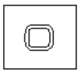
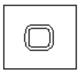
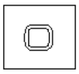
In the PRG STATE page, before the inputted words will be executed, if there is an error in inputted words, press CANCEL to cancel highligt state, then program segment can be modified. It may press RESET key to clear all the words, then input the correct words; for example ,"Z1000" will be inputted to replace Z100 in Section 5.1 of this chapter, the steps are as follow.

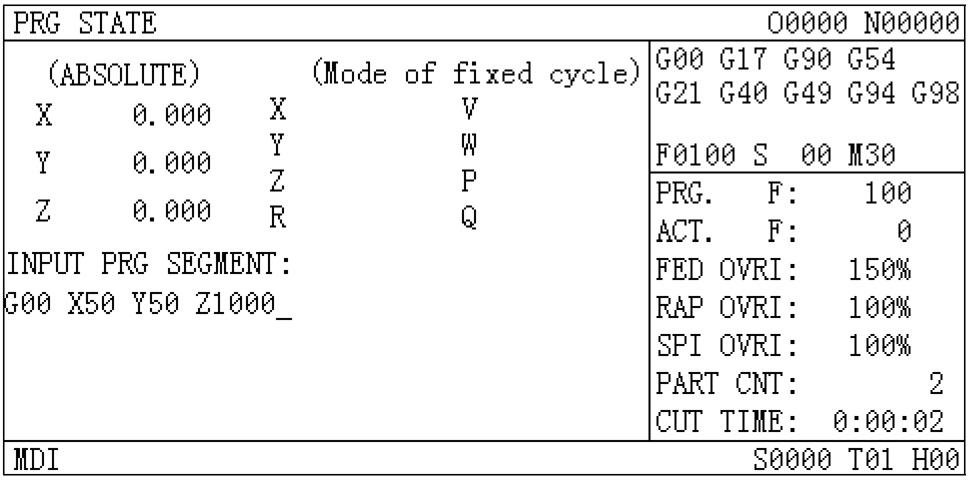
1. press CANCEL key，the page is as follows：



2. press CANCEL key，the page is as follows：

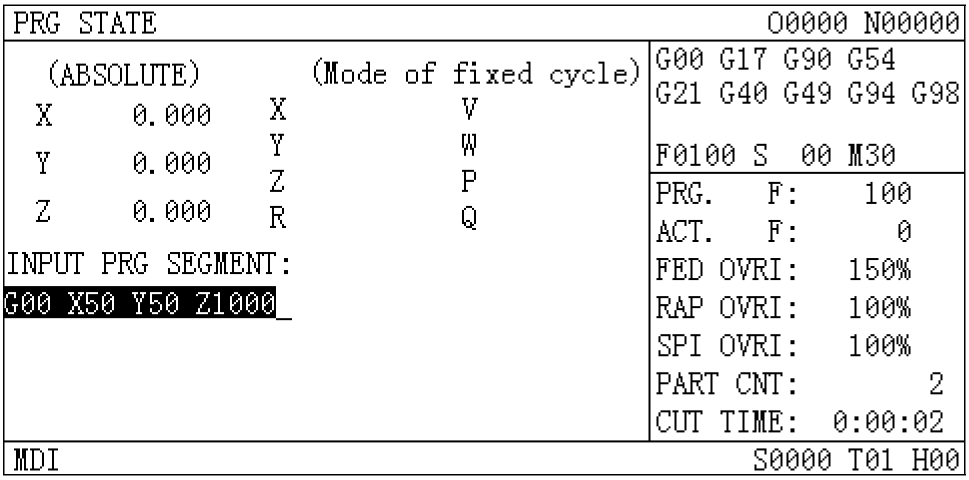


3. press . . . .  by sequence, the page is as follows:



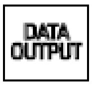
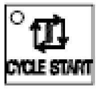
4. At last ,press 

, the page is as follows:



5.5 OUT Key Start

When the "OUTR" of the K parameter K0010 is set to 1, the current words inputted

may be executed by pressing  key in MDI mode. It is the same as .

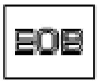
CHAPTER 6 PROGRAM EDIT AND MANAGEMENT

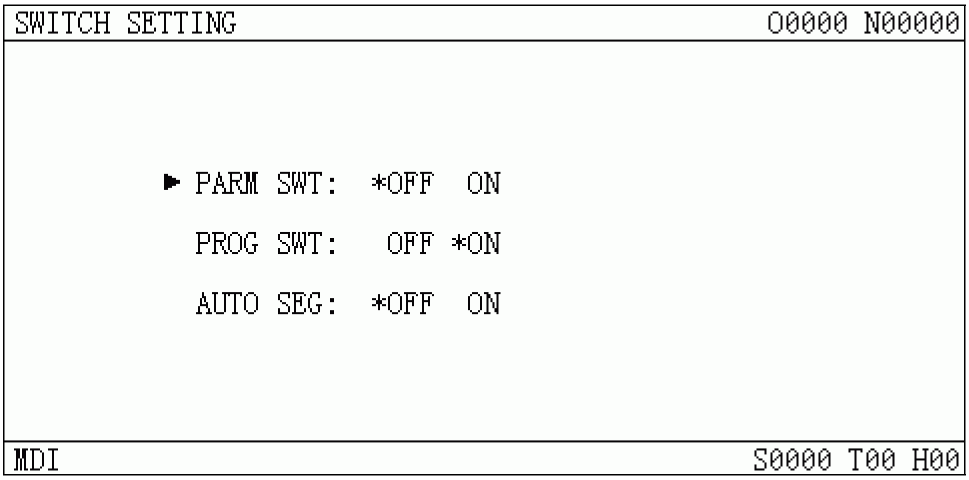
In Edit mode, the programs can be created, selected, modified, copied and deleted, and the bidirectional communication between CNC and CNC, or CNC and PC can also be achieved. To prevent the program to be modified or deleted accidentally, a program switch is set for this GSK980MD system. And it must be turned on before program editing. Also 3 level user authority is set in this GSK980MD system to facilitate the management. Only the operation authority is above 4 level (4 or 3 level etc.) can open the program switch for program editing.

6.1 Program Creation

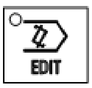
6.1.1 Creation of the block number

The program can be with or without a block No. The program is executed by the block numbered sequence (except the calling). When the “AUTO SEG”switch in setting page is OFF, the CNC doesn’t generate the block number automatically, but the blocks may be edited manually.

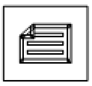
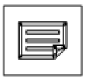
When “AUTO SEG” switch in switch setting page is on, the CNC generates the block number automatically. In editing, press  key to generate block number of the next block automatically. The increment of this block number is set by №216.



6.1.2 Input of the program content

1 Press  key to enter the Edit mode;

2 Press  key to enter the Program interface, select the PRG CONTENT page

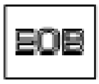
by pressing  or  key

```
PRG CONTENT                SEG1       COL:1     C:/O0000.CNC
O0000 (O0000);
G0 G54 G90 X0 Y0 Z0;
X10 Y10;
X-10 Y-10;
M99;
%

EDIT                                            S0000 T00 H00
```

3 Key in address key O, numerical key 0, 0, 0 and 1 key by sequence (e.g. Program O0001 creation);

```
PRG CONTENT                SEG1       COL:1     C:/O0000.CNC
O0000 (O0000);
G0 G54 G90 X0 Y0 Z0;
X10 Y10;
X-10 Y-10;
M99;
%

    O0001_
EDIT                                            S0000 T00 H00
```

4 Press EOB key to setup the new program;

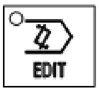
5 Input the edited part program one by one, the character will be displayed on the screen immediately as it is input(as for compound key, press this key repeatedly for alternate input),after a block is finished, press 

to terminate it.

6 Other blocks cab be input by step 5 above.

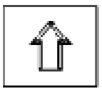
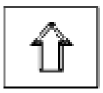
6.1.3 Search of the character

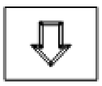
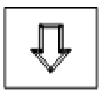
1 Scanning: To scan the character one by one by cursor

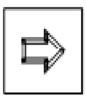
Press 

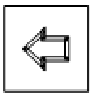
key to enter the Edit mode, then press 

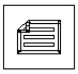
key to enter the PRG CONTENT page;

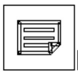
1）Press  key, the cursor shifts a line upward; if the number of the column where the cursor locates is over the total columns of the previous line, the cursor moves to the previous block end (at";"sign) after  key is pressed;

2）Press  key, the cursor shifts a line downward; if the number of the column where the cursor locates is over the total columns of the next line, the cursor moves to the next block end (at";"sign) after the  key is pressed;

3） Press  key, the cursor shifts a column to the right; if the cursor locates at the line end, it moves to the head of the next block;

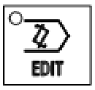
4) Press  key, the cursor shifts a column to the left; if the cursor locates at the line head, it moves to the end of the next block;

5) Press key to page upward, the cursor moves to the 1st line and the 1st column of the previous page, if it pages to the head of the program, the cursor moves to the 2nd line and 1st column;

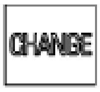
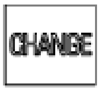
6) Press key to page downward, the cursor moves to the 1st line and 1st column of the next page, if it pages to the end of the program, the cursor moves to the last line and 1st column of the program;

2 Searching: To search for the specified character upward or downward from the cursor current location

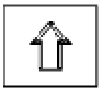
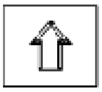
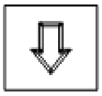
The steps of searching are as follows:

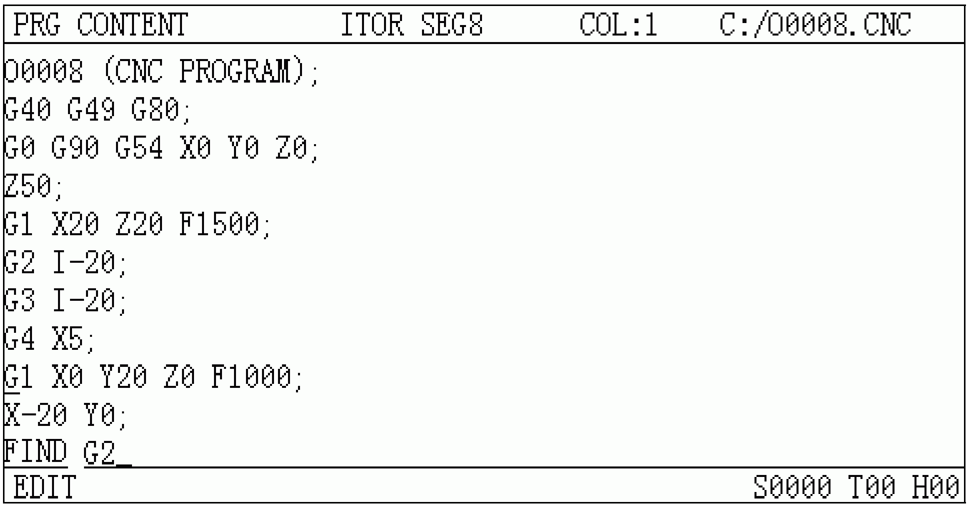
1) Press  key to enter Edit mode;

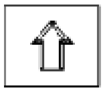
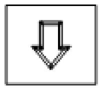
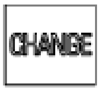
2) Press  key to enter the PRG CONTENT page;

3)Press  key to enter Search mode, Max. 50 bytes can be input, but only 10 of them can be searched. If the characters are over 10 bytes, searching will fail. E.g. to search command ——G2, press  key, then input G2, and operate as step 4.

```
PRG CONTENT          ITOR SEG8      COL:1    C:/O0008.CNC
O0008 (CNC PROGRAM);
G40 G49 G80;
G0 G90 G54 X0 Y0 Z0;
Z50;
G1 X20 Z20 F1500;
G2 I-20;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;
FIND G2_
EDIT                                         S0000 T00 H00
```

4）Press  key（  or  by the location relation between the character to be searched and the character where the cursor locates), it displays as follows:



5）After the searching, the CNC system is still in searching state, press  or  key again, the next character can be searched. Or press  key to exit the searching state.

6）If the character is not found, the prompt of “Srch fail” will be displayed.

Note:During the searching, it doesn’t search the characters in the called subprogram

3 Method to return to the program head

1) In the Program Display page of the Edit mode, press  key, the cursor returns to the program head

2) Search the program head character by the methods in Section 6.1.3 of this part.

6.1.4 Insertion of the character

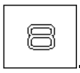
Steps:

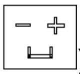
1）Select the PRG CONTENT page in Edit mode, the page is as follows:

```
PRG CONTENT                  SEG5       COL:1     C:/O0008.CNC
O0008 (CNC PROGRAM);
G40 G49 G80;
G0 G90 G54 X0 Y0 Z0;
Z50;
G1 X20 Z20 F1500;
G2 I-20;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;

EDIT                                          S0000 T00 H00
```

2）Input the character to be inserted(to insert G98 code before G2 in the above figure, input 

. 

. 

. 

), the page is as follows:

```
PRG CONTENT                  SEG5       COL:5     C:/O0008.CNC
O0008 (CNC PROGRAM);
G40 G49 G80;
G0 G90 G54 X0 Y0 Z0;
Z50;
G1 X20 Z20 F1500;
G98 G2 I-20;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;

EDIT                                          S0000 T00 H00
```

Note 1:In the Insert mode, if the cursor is not located at the line head, a space will be automatically generated when inserting the command address; if the cursor is located at the line head, the space will not be generated, and it should be inserted manually.

Note 2: In program content edit mode or MDI mode of program state page，press 

key to enter insertion or macro edit state.

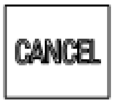
In macro editting mode，special symbols can be input are: ‘[’. ‘]’. ‘=’. ‘+’. ‘>’. ‘<’. ‘/’. ‘&’. ‘|’. **Above symbols are frequently used for macro edit.**

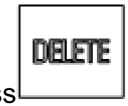
Difference between two states	**Automatic space**	**Process of character ‘O’**	**Input special symbols**
Insertion state	In program editting, insert blank automatically to separate words.	Program switch, duplication and deletion can be done by pressing ‘O’.	Special symbols can not be inputted.
Macro edit state	Blank can not be inserted automatically.	Only input character ‘O’.	Special symbols can be inputted.

6.1.5 Deletion of the character

Steps:

1）Select the PRG CONTENT page in Edit mode;

2）Press 

key to delete the character before the cursor; press 

key to delete the character where the cursor locates.

6.1.6 Modification of the character

Cancel or delete the character and re-enter new ones.

6.1.7 Deletion of a single block

This function is only applied to the block with a block No.(N command) , which is at the head of a line and followed by blocks which are divided by space.

Steps:

1）Select the PRG CONTENT page in Edit mode;

2）Move the cursor to the head of the block to be deleted (column 1— where N locates), then press 

key.

Note: If the block has no block No.N, key in “N”at the head of the block, and move the cursor to “N”, then press 

key.

6.1.8 Deletion of the blocks

It deletes all the content (including the specified block)from the current character where the cursor locates to the block with the specified No.(searching downward), and the

specified block must has a block No..

```
PRG CONTENT                    SEG5       COL:5     C:/O0008.CNC
O0008 (CNC PROGRAM);
G40 G49 G80;
G0 G90 G54 X0 Y0 Z0;
Z50;
G1 X20 Z20 F1500;
N10 G98 G2 I-20;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;

EDIT                                            S0000 T00 H00
```

Steps

1) Select the PRG CONTENT page in Edit mode;

2) Press CANCEL key to enter the FIND state, and key in the block No.

```
PRG CONTENT                    SEG2       COL:1     C:/O0008.CNC
O0008 (CNC PROGRAM);
G40 G49 G80;
G0 G90 G54 X0 Y0 Z0;
Z50;
G1 X20 Z20 F1500;
N10 G98 G2 I-20;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;
FIND N10_
EDIT                                            S0000 T00 H00
```

3) Press DELETE key to delete blocks from G0 (block 2) to N10 (including block N10). It displays as follows:

```
PRG CONTENT                    SEG2       COL:1     C:/O0008.CNC
O0008 (CNC PROGRAM);
G40 G49 G80;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;
X0 Y-20 Z-10;
X20 Y0 Z-20;
X5 Y5 Z-50;
M99;

EDIT                                            S0000 T00 H00
```

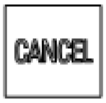
6.1.9 Segment deletion

It deletes the content downward from the current character where the cursor locates to the word specified.

```
PRG CONTENT                SEG2      COL:4    C:/O0008.CNC
O0008 (CNC PROGRAM);
G40 G49 G80;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;
X0 Y-20 Z-10;
X20 Y0 Z-20;
X5 Y5 Z-50;
M99;

EDIT                                         S0000 T00 H00
```

Steps

1）Select the PRG CONTENT page in Edit mode

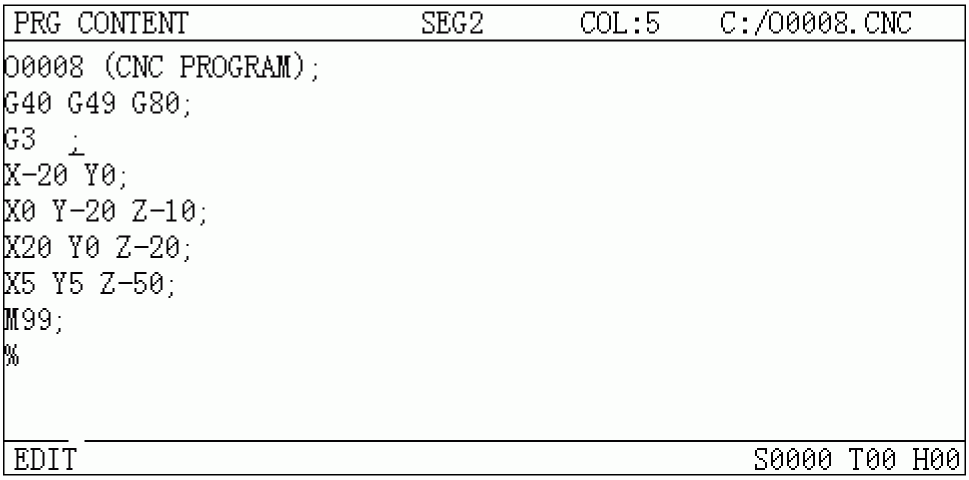
2）Press 

key to enter the FIND state, and key in the characters (see the following figure: input F1000)

```
PRG CONTENT                SEG2      COL:4    C:/O0008.CNC
O0008 (CNC PROGRAM);
G40 G49 G80;
G3 I-20;
G4 X5;
G1 X0 Y20 Z0 F1000;
X-20 Y0;
X0 Y-20 Z-10;
X20 Y0 Z-20;
X5 Y5 Z-50;
M99;
FIND F1000_
EDIT                                         S0000 T00 H00
```

3）Press 

key, and all programs from I-20 where the cursor locates to F1000. It displays as follows:



Note 1:If the specified character is not found or the specified character is located before the current cursor, the prompt of "Srch fail" will be displayed. If there are multiple same characters specified downward, it defaults the nearest one to the current cursor.

Note 2: If the command address is input, both the address and the command value behind it are Deleted.

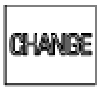
6.2 Program annotation

To facilitate the user to search, manage and edit program, the system provides program name annotation and block annotation functions.

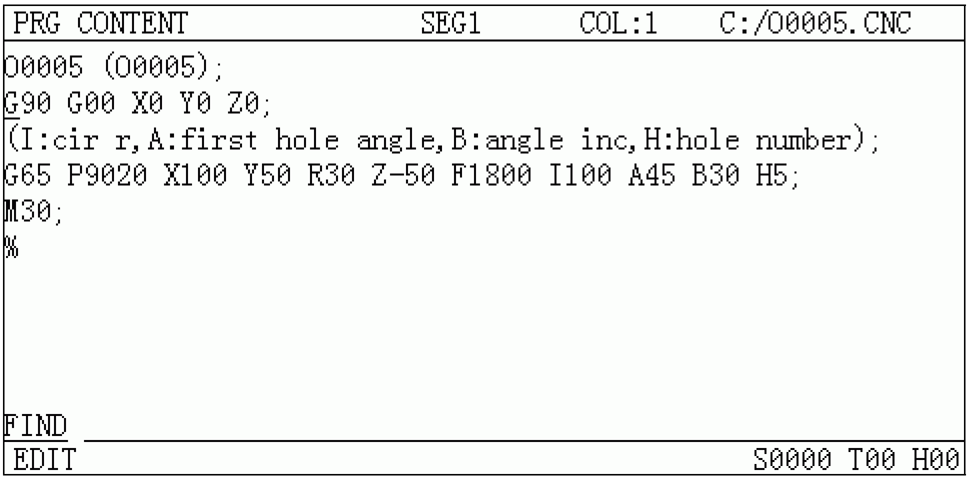
6.2.1 Annotation for program name

The program annotation can be added in the brackets behind it. For exa mple: program O0005 is used for machining bolt holes, the annotation can be added in program contents as follows:

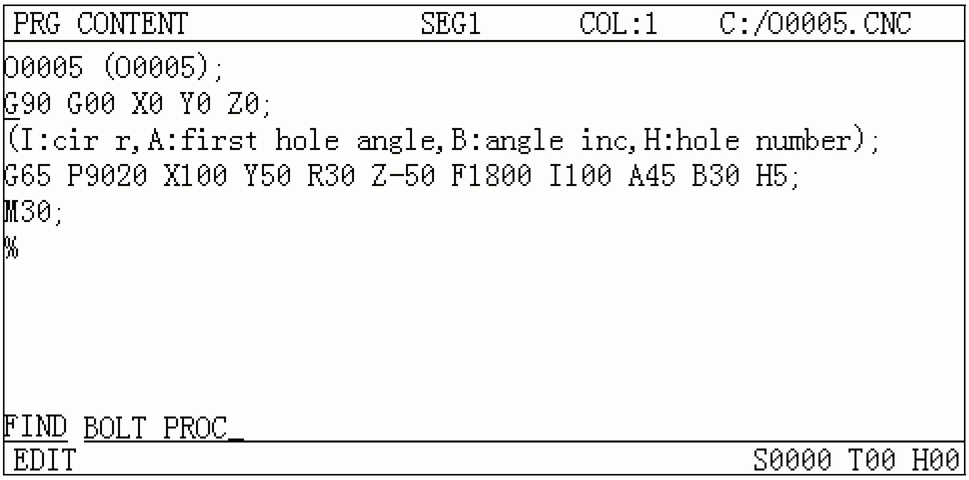
1）Select edit mode, and then enter program content display page.

2）Press 

key，search is displayed at the left bottom of the screen, the displayed figure is as follows:

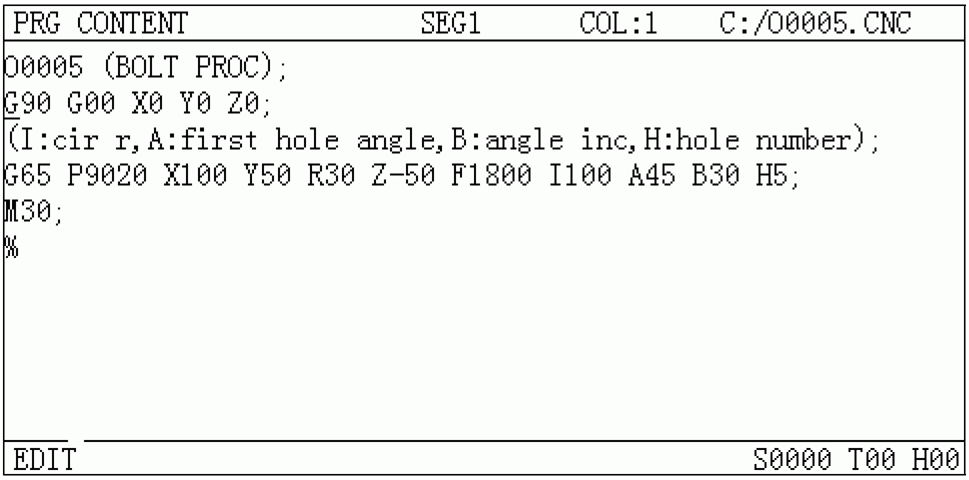


3）Input annotation behind search (input max. 50 characters except for brackets). If BOLT PROC is inputted (bolt holes machining), the page displayed is as follows:



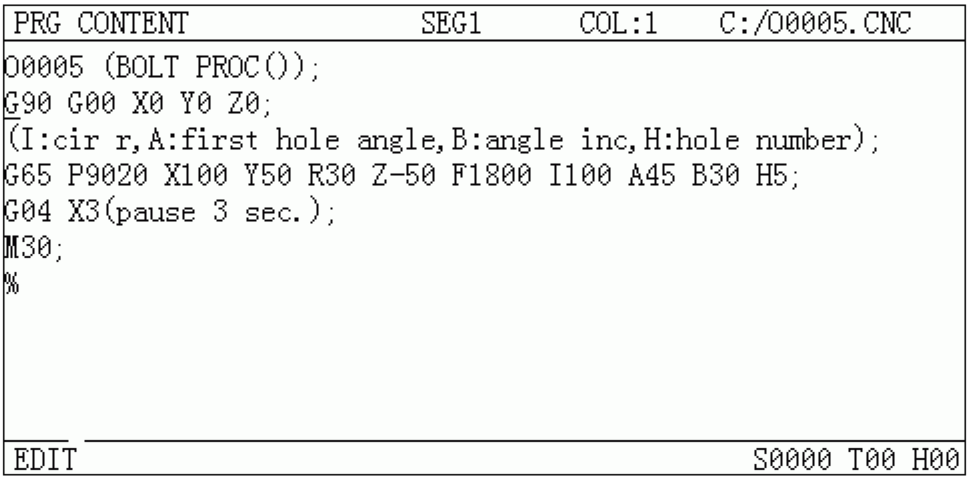
4）Press 

key，program annotation setting up is finished，the displayed page is as follows：



6.2.2 Block annotation

Take contents in brackets '（'and'）'as program annotation, which can be put at any position of a block and displayed with green characters. The page is as follows:



Related explanations：

1）Because symbols'（'and '）'are not provided in the system，block annotation can not be inputted by edit mode in the system. If block annotation is needed to added, edit annotation on the PC and download it to the CNC by software.

2）The system is not support Chinese characters. If Chinese characters are edited on PC, which will be displayed as blanks in the system after it is saved in the CNC.

Note 1: After a program is set up, if the program name annotation is not added, CNC defaults program name as program name annotation

Note 2: Program annotation in the CNC must be English, but the CNC supports Chinese annotation display (except for Chinese decimal points). The way of adding Chinese annotation is as follows: Edit Chinese annotation in the PC machine, and then download it to the CNC by communication software.

6.2.3 Alter program annotation

Operation steps are the same as program annotation setting steps on section 6.2.1 of this chapter.

6.3 Deletion of the Program

6.3.1 Deletion a single program

Steps:

1）Select the PRG DISPLAY page in Edit mode;

2）Key in address key O, numerical key 0. 0. 0. 1 by sequence(take program O0001 for an example**);**

3) Press key, program O0001 will be deleted

Note: Press 'DELETE ' key in page 'program preview'or'file list'to delete program.

6.3.2 Deletion of all programs

Steps

1）Select the PRG DISPLAY page in Edit mode

2）Key in address key O , symbol key - + numerical key 9. 9. 9. 9 by sequence

3）Press DELETE key, all the programs will be deleted.

Note: Press 'delete key'in page 'file list'to delete all programs.

6.4 Selection of the Program

When there are multiple programs in CNC system, a program can be selected by the following 4 methods:

6.4.1 Search method

1）Select Edit mode;

2）Press PROGRAM key to enter the PRG CONTENT page;

3）Press address key O and key in the program No.;

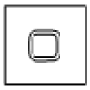
4）Press ⇩ or EOB key, the searched program will be displayed.

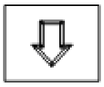
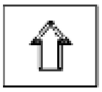
Note:In step 4, if the program does not exist, a new program will be created by CNC system after EOB key is pressed

6.4.2 Scanning method

1）Select Edit or Auto mode;

2）Press  key to enter the PRG DISPLAY page;

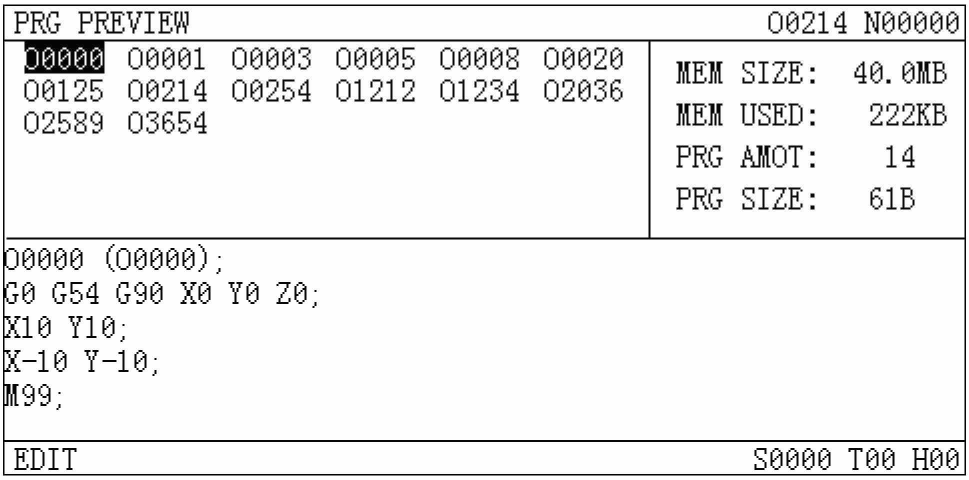
3）Press address key 

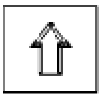
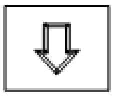
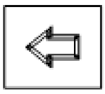
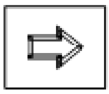
4）Press  or  key to display the next or previous program;

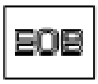
5）Repeat step 3 and 4 to display the saved programs one by one.

6.4.3 Cursor method

1）In Program Preview mode **(must be in non-running state)**;

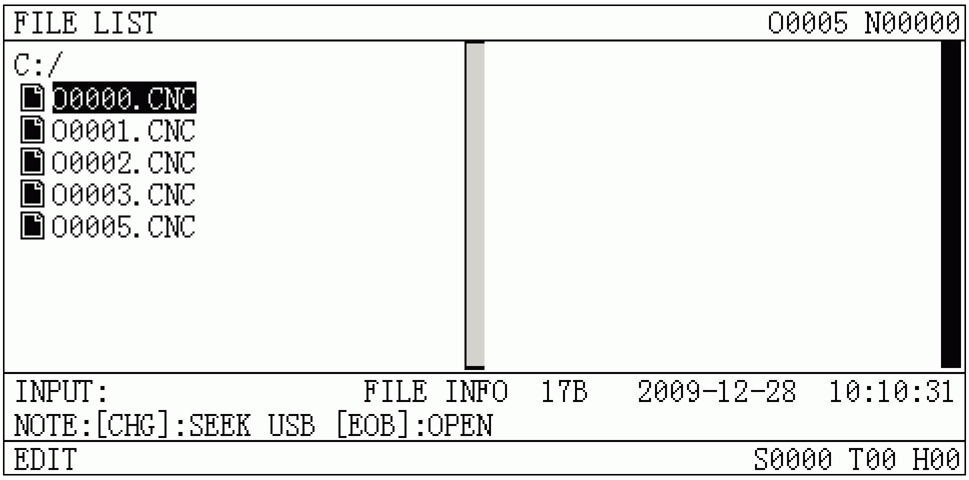


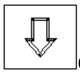
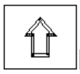
2）Press  .  .  or  key to move the cursor to the program name to be selected (change “PRG SIZE”, “NOTE” content as the cursor moves);

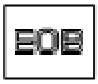
3）Press  to open the program.

6.4.4 Select file by using file list

1） On file list page（Edit mode is operation mode）



2）Select program to be opened by pressing  or  key.

3) Open program by pressing 

key.

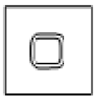
6.5 Execution of the Program

After the program to be executed is selected by the method in Section 6.4 of this part, select the Auto mode, then press 

key (or press external cycle start key), the program will be executed automatically.

6.6 Rename of the Program

1）Select the PRG CONTENT page in Edit mode;

2）Press address key  and key in the new program name;

3）Press 

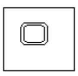
key.

Note: No matter whether the program is altered or not, program annotation is changed into new program name automatically after program is renamed.

6.7 Copy of the Program

To save the current program to a location:

1）Select the PRG CONTENT page in Edit mode;

2）Press address key  and key in the new program No

3）Press CHANGE key.

6.8 Program positioning

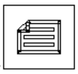
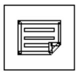
- To the position where the program stops last time by TO

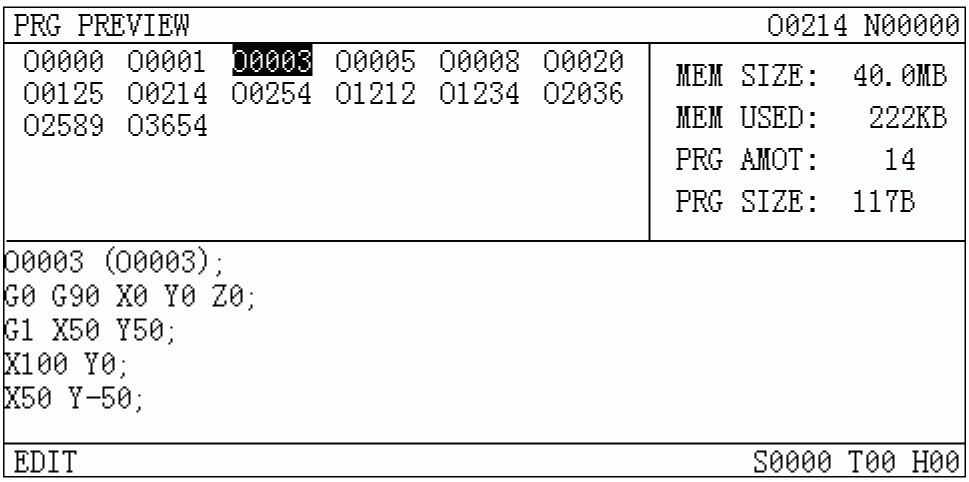
  Search for the point where the program execution stops by TO. Select edit mode to enter program content page and press conversion key, input TO to search which is displayed at the left bottom. Then press up or down key, searching and positioning are displayed at this time, the cursor will move to the position where program stops last time.

- Position to specified block by TO＋num（num is the block number specified by user. For example: TO10000 means position to the 10000th block）

  On program content page, locate to specified block by inputing TO block number. Press conversion key after entering program content page, input TO to search which is displayed at the left bottom and then press up or down key, the cursor will move to the specified program.

6.9 Program preview

In non-edit mode，press PROGRAM key to enter program preview page. In this page, program names saved in CNC are displayed in the form of list. Max. 36 program names can be displayed In one page, if programs saved are over 36, press   CHANGE key to display programs in other page.



- Program capacity display：

On top right window, “storage capacity”displays the max. capacity of program which can be saved in CNC. “Used capacity”displays the capacity of saved program in CNC system.. “Program

number"displays the program number saved in the CNC system. "Program size"displays the size of the currently opened program.

- Program preview selection：

On top left of the window, the name of currently previewed program will be displayed in blue characters on white ground. Program size on top left window is the size of currently previewed program. The following window displays currently previewed progam, display 5-line program.

- Usage of cursor key and conversion key：

When select program in a program list, select the program to be previewed by cursor moving key on MDI panel. If the size is very big, max. 36 program names can be displayed in program list. Select program by pressing right moving key or pressing conversion key directly, turn pages to display the program list, and then select it by cursor moving key on MDI panel.

- Open a program：

In edit, auto, MDI modes, when open the program on program preview window, this program can be opened by pressing EOB key on MDI panel. At the same time, the name of currently opened program is displayed on top right page.

- Deletion of program

Move cursor to the program will be deleted, press delete key and then press Y key or N key on multiple select manue to select wether delete it or not

CHAPTER 7 AUTO OPERATION

Note!

The keys functions of this 980MDa machine panel are defined by Ladder; please refer to the respective materials by the machine builder for the function significance.

Please note that the following function introduction is described based on the

7.1Auto Run

7.1.1 Selection of the program to be run

1. Search method

1）Select the Edit or Auto mode;

2）Press PROGRAM key to enter the PRG CONTENT page;

3）Press the address key O and key in the program No.

4）Press ⇧ or ⇩ key, the program retrieved will be shown on the screen, if the program doesn't exist an alarm will be issued

Note In step 4, if the program to be retrieved does not exist, a new program will be setup by CNC system after pressing EOB key.

2 Scanning method

1）Select the Edit or Auto mode

2） Press PROGRAM key to enter the PRG display page

3）Press the address key O

4）Press the ⇧ or ⇩ key to display the next or previous program;

5）Repeat the step 3, 4 above to display the saved program one by one.

3 Cursor method

a) Select the Auto mode (must in non-run state)

b) Press key to enter the PRG LIST page;

c) Press . . . key to move the cursor to the name of the program to be selected;

d) Press key.

4. File open method

Select the edit or operation mode:

1）Press key twice to enter the page of file list.；

2）Press , keys to move the cursor to the file will be selected.

3）Press key to select a file.

4）Press key to open the selected file.

Note: The file can not be opened if the expanded name is not“.CNC”.

7.1.2 Program start

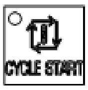
1. Press key to select the Auto mode

2. Press key to start the program, and the program execution begins

Note Since the program execution begins from the block where the cursor locates, before pressing the key, make a check whether the cursor is located at the block to be executed. If begins from the start line, but the cursor is not in this line, move the cursor to the line.

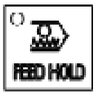
7.1.3 Stop of the auto run

- **Stop by command (M00)**

the block containing M00 is executed, the auto run is stopped. So the modal function and state are all reserved. Press the key 

or the external Run key, the program execution continues.

- **Stop by a relevant key**

1 In Auto run, by pressing key 

or external dwell key, the machine remains at the following state:

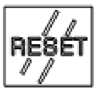
（1）The machine feed decelerate to stop;

（2）During the execution of the dwell command (G04), it pauses after G04 command execution is finished.

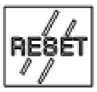
（3）The modal function and state are saved;

（4）The program execution continues after pressing the 

key

2 Stop by Reset key 

（1）All axes movement is stopped.

（2）M, S function output is invalid (the automatic cut-off of signals such as spindle CCW/CW, lubrication, cooling by pressing 

key can be set by the parameters)

（3）Modal function and state is held on after the auto run.

3 Stop by Emergency stop button

If the external emergency button (external emergency signal valid) is pressed under the dangerous or emergent situation during the machine running, the CNC system enters into emergency state, and the machine moving is stopped immediately, all the output (such as spindle rotation, coolant) are cut off. If the Emergency button is released, the alarm is cancelled and CNC system enters into reset mode.

4 By Mode switching

When the Auto mode is switched to the Machine zero, MPG/Step, the current block “dwells”immediately; when the Auto mode is switched to the Edit, MDI mode, the “dwell”is not displayed till the current block is executed.

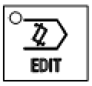
Note 1 Ensure that the fault has been resolved before cancelling the emergency alarm.

Note 2　The electric shock to the device may be decreased by pressing the Emergency button before power on and off.

Note 3　The Machine zero return operation should be performed again after the emergency alarm is cancelled to ensure the the coordinate correctness (but this operation is unallowed if there is no machine zero in the machine)

Note 4　Only the BIT3 (ESP) of the bit parameter No.017 is set to 0, could the external emergency stop be valid.

7.1.4 Auto run from an arbitrary block

1. Press 

key to enter the Edit mode, press 

key to enter the Program interface, or press 

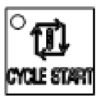
key several times to select the PRG CONTENT page:

2. Move the cursor to the block to be executed (for example, move the cursor to the 3th line head if it executes from the 3th line);

```
PRG CONTENT              SEG3      COL:1    C:/O0000.CNC
O0000 (O0000);
G0 G54 G90 X0 Y0 Z0 G49;
G01 X100 Y100 F500;
G02 I20;
G01 X52 Z01;
G91 X2 Z-6.3;
G00 X0 Y0 Z0;
M30;
%

EDIT                                      S0000 T00 H00
```

3. If the mode （G, M, T, F command）of the current block where the cursor locates is defaulted and inconsistent with the running mode of this block, the corresponding modal function should be executed to continue the next step.

4. Press 

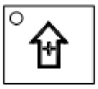
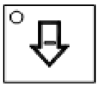
key to enter the Auto mode, then press 

key to start the program.

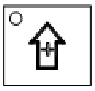
7.1.5 Adjustment of the feedrate override, rapid override

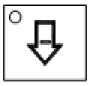
In Auto mode, the running speed can be altered by adjusting the feedrate override, rapid override with no need to change the settings of the program and parameter.

- Adjustment of the feedrate override

Press the  or  key in 

, it can realize 16-level real time feedrate adjustment.

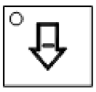
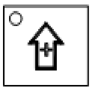
Press the  key each time, the feedrate override ascends a gear level till 150%

Press the  key each time, the feedrate override decends a gear level till 0;

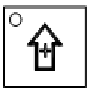
Note 1 The actual feedrate value is specified by F in program feedrate override adjustment;

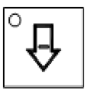
Note 2 Actual feedrate= value specified by F× feedrate override

- Adjustment of rapid override

It can realize the 4-level real time rapid override Fo. 25％. 50％. 100％ adjustment by pressing the  or  key in 

.

Press the  key each time, the rapid override ascends a level till 100%;

Press the  key each time, the rapid override decends a level till F0

Note 1 The max. rapid traverse speeds of X, Y, Z axis are set by bit parameter No.059, No.060, No.061 respectively;

X axis actual rapid traverse rate = value set by parameter No.059×rapid override

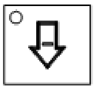
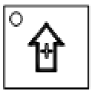
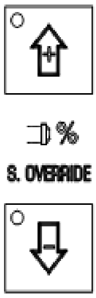
Y axis actual rapid traverse rate = value set by parameter No.060×rapid override

Z axis actual rapid traverse rate = value set by parameter No.061×rapid override

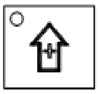
Note 2　When the rapid override is F0, the rapid traverse rate is set by bit parameter No.069.

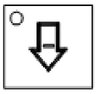
7.1.6 Spindle override adjustment

While the spindle speed is controlled by the analog voltage output in Auto mode, it can be adjusted by spindle override.

Press the  or  key in 

to adjust the spindle override for the spindle speed, it can realize　8-level real-time override adjustment between 50％～120％.

Press the  key each time, the feedrate override ascends a level till 120%

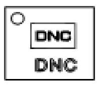
Press the  key each time, the rapid override decends a level till 50%.

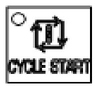
Note 1　The actual output analog voltage=analog voltage by parameter×spindle override

Example:　When the bit parameter No.101 is set to 9999, No.100 to 645, execute S9999　command　to select the spindle override 70%, the actual output analog voltage≈10×70%=7V

7.2 DNC running

This CNC system has a DNC function, by the connection of the DNC communication software with this system, the high speed, high capacity program can be performed in this system.

In Auto mode, press the 

key, it enters the DNC mode. Then press the 

key to start the program　DNC machining under the condition that the PC is get ready

Please refer to the DNC communication software for details.

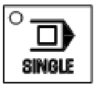
7.3 Running state

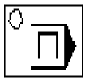
7.3.1　Single block execution

When the program is to be executed for the 1st time, to avoid the programming errors, it may select Single block mode to execute the program.

In Auto mode, the methods for turning on single are as follows.

Volume II Operation

Press the 

key to make the single block indicator  in State area to light up, it means that the single block function has been selected

In Single block mode, when the current block execution is finished , the CNC system stops;if next block is to be executed,it needs to press the 

key.

Note Even at the mid point, the single block stops in G28,G29, G30 commands

7.3.2 Dry run

Before the program is to be executed, in order to avoid the programming errors, it may select the Dry run mode to check the program. And the machine runs by a constant speed other than the speed specified by the program.

In Auto mode, the method for turning on the Dry run switch are as follows.

Press 

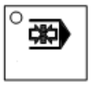
key to make the dry run indicator in State area to light up, it means that the dry run function is selected

The speed specified by the program is invalid in Dry run, and actural feedrate is set by the DATA parameter No.174.

7.3.3 Machine lock

In Auto mode, the ways to make machine lock function valid are as follows.

Press the 

key to make the machine lock indicator  in State area to light up, it means that it has enterd the machine lock state.

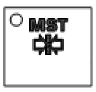
While in the machine lock mode:

1. The machine carriage doesn't move, the “MACHINE”in the INTEGRATED POS page of the POSITION interface doesnt' vary too. The RELATIVE POS and ABSOLUTE POS, DIST TO GO are refreshed normally
2. M, S, T commands can be executed normally.

7.3.4 MST lock

In Auto mode, the ways to make MST lock function valid are as follows.

Press the 

key to make the MST lock indicato 

in State area to light up, it means that it has entered the MST lock state. And the carriage move is not performed by M, S, T

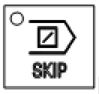
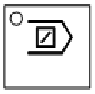
commands

Note: When the MST lock is valid, it has no effect on the execution of M00, M30, M98,M99.

7.3.5 Block skip

If a block in program is not needed to be executed and not to be deleted, this block skip function can be used. When the block is headed with “/”sign and Block skip function is valid, this block is skipped without execution in Auto mode

In Auto mode, the way to make block skip function valid is as follows.

Press the  key to make the block skip indicator  in State area to light up, it means that the block skip function is valid.

Note While the block skip function is invalid, the blocks headed with “/”signs are executed normally in Auto mode.

7.3.6 Optional stop

In AUTO mode, the valid optional stop function is as follows:

Press  key to enter optional stop and the indicator lights up.

The program will be “stopped” at command M01. Press  key again to continue program execution.

7.4 Memorizing at power-down

7.4.1 Program interruption in non-DNC auto operation

Operation method 1 (Manual)

1. After power on, press conversion key →press letter “T”+letter“O”→up, down moving keys on pages“program content, edit” to the block where the execution stops last time.
2. Switch to the pages “coordinate & program, machine zero”.
3. Enter the next step after machine zero is performed.
4. Switch to manual or MDI mode. Locate to the block where it stops last time. (At this moment, it is necessary to confirm whether it is at state G40, G49, G54. Ensure that tools are in a safe range during positioning.)
5. Switch to manual mode, press conversion key. It prompts “Locate to the block where it stops last time. It will recover the mode before power-down（Y/N）”.
6. Press Y to recover the mode before power-down.
7. Switch to auto mode, press cycle start key to execute the block continuously from where it stops last time.

Operation method 2 (Auto)

1. After power on, press conversion key →press letter “T”+letter“O”→up, down moving keys on pages“program content, edit” to the block where the execution stops last time.
2. Switch to the pages “coordinate & program, machine zero”.
3. Perform machine zero operation.
4. After machine zero is performed, press conversion key. It prompts at the bottom of the screen: “Locate to the block automatically where it stops last time. It will recover the mode before power-down（Y/N）”. Input Y (Ensure that tools moving path is in a safe range at this moment.). Coordinates start move, it locates to the block where it stops last time, and recovers the mode before power-down.
5. Switch to auto mode, press cycle start key to execute the block continuously where it stops last time.

7.4.2 Interruption at power-down on DNC auto operation

Operation method (Auto)

1. Switch to “coordinate program, machine zero return” after power on.
2. Execute machine zero return.
3. After machine zero return is finished, press conversion key. It prompts at the bottom of the screen: “Locate to the block automatically where it stops last time. It will recover the mode before power-down（Y/N）”. Input Y (Make sure tools moving path is in a safe range at this moment.). Coordinates start move, it locates to the block where it stops last time, and recovers the mode before power-down.
4. Switch to the highlighted block when DNC, CNC power down.
5. Search for the interrupted block in DNC transmission software, then press RESET key on panel to continue PC software transmission. Press cycle start key to continue execution.

CHAPTER 8 MACHINE ZERO RETURN OPERATION

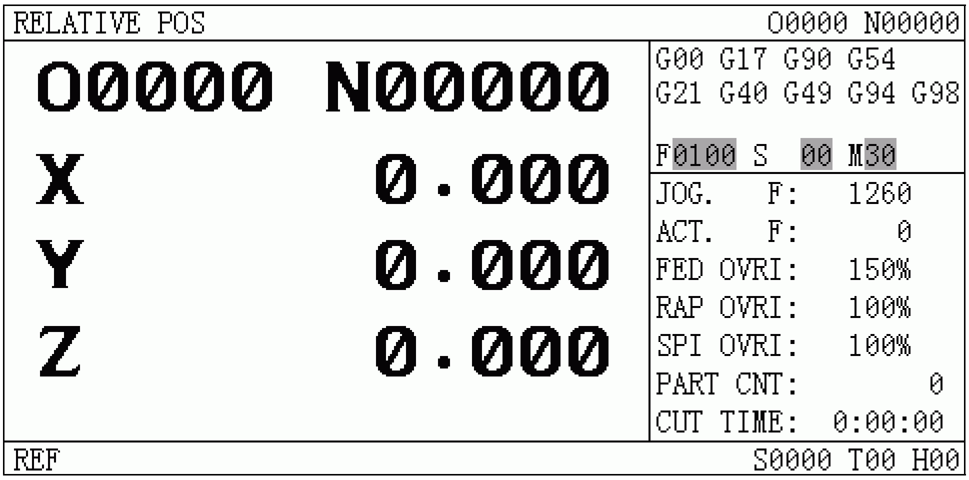
8.1 Machine Zero

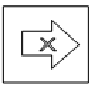
The machine coordinate system is a basic coordinate system for CNC coordinate calculation. It is an inherent coordinate system of the machine. The origin of the machine coordinate system is called **machine zero** (or mechanical reference point). It is defined by the zero return switches fixed on the machine. Usually the switch is fixed on the positive max. Strokes of X, Y, Z axes.

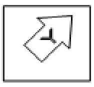
8.2Machine Zero Return Steps

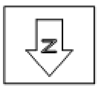
1 Press 

key, it enters the Machine zero mode, the bottom line of the screen page shows "REF", the figure is as follows:



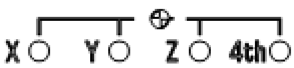
2 Press 

or 

or 

key to select the machine zero of X, Y or Z axis

3 The machine moves along the machine zero direction, and returns to the machine zero via the deceleration signal, zero signal detection. And the axis stops with the machine zero finish indicator lighting up.



Machine zero finish indicators

Note1：If the machine zero is not fixed on the machine, machine zero operation B/C/D is unallowed.

Note2：While the coordinate is moved out from the machine zero, the machine zero finish indicators go out.

Note3：After the machine zero operation, the cancellation of the tool length offset for the

CNC is set by the BIT7 of the bit parameter No.22

Note4：See details in the 3rd part INSTALLATION AND CONNECTION for the parameters concerning with the machine zero.

Note 5: When machine zero return, bit parameter №011 ZNIK determines whether axis movement is locked automatically.

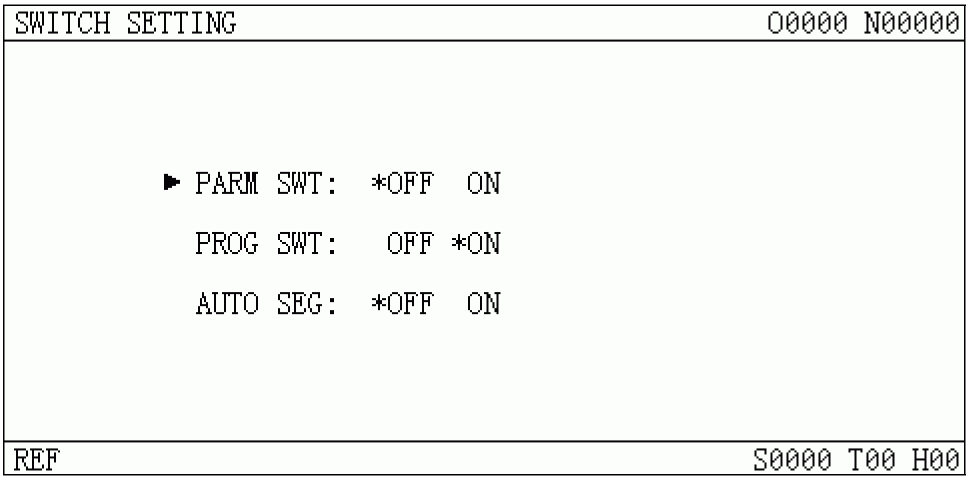
Note 6: Only machine zero D mode can be used for rotary axis.

CHAPTER 9　DATA SETTING, BACKUP and RESTORE

9.1Data Setting

9.1.1 Switch setting

In SWITCH SETTING page, the ON-OFF state of PARM SWT (parameter switch), PROG SWT (program switch), AUTO SEG (auto sequence No.) can be displayed and set, the figure is as follows:



1 Press key to enter the Setting interface, then press or key to enter SWITCH SETTING page

2 Press or key to move the cursor to the item to be set

3 Press . and . key to shift the ON-OFF state, press or key, "*"moves to the left to set the switch for OFF, Press or key, "*"moves to the right to set the switch for ON.

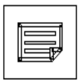
Only the PARM SWT is set to ON, could the parameter be altered; so are PROG SWT and AUTO SEG

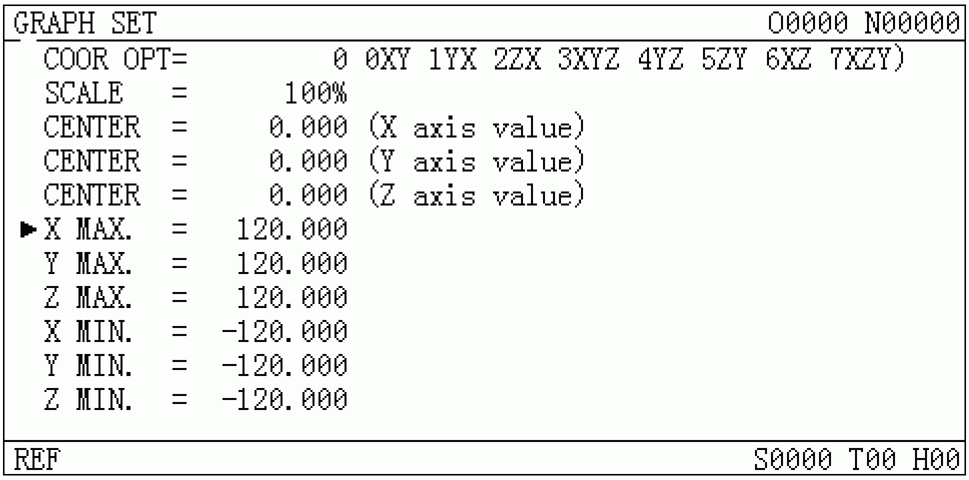
Note 1: When parameter switch is shifted from "off"to"on"for the first time, CNC alarm occurs. Press , keys together to eliminate the alarm. Alarm will not occur when parameter switch is shifted again. For security, set parameter switch to "off" after parameter alteration is finished.

Note 2: When parameter switch is shifted from "off"to"on", CNC alarm occurs. Alarm will occur again when parameter switch is shifted from "on"to"off"for the first time. Press , keys together to eliminate the alarm.

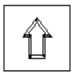
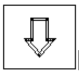
9.1.2 Graphic setting

Press 

key to enter graphic interface. Press  or  key to access the following graphic parameter page.



A：The way of setting graphic parameter

1. In MDI mode, press  or key to move the cursor to the parameter to be set,
2. Input corresponding valus,
3. Press

key，and the setting is finished.

B：Significance of graphic parameter

Coordinate selection: Display view angle of the graphic path can be selected by setting different values. Corresponding coordinate for 0~7is as follows.

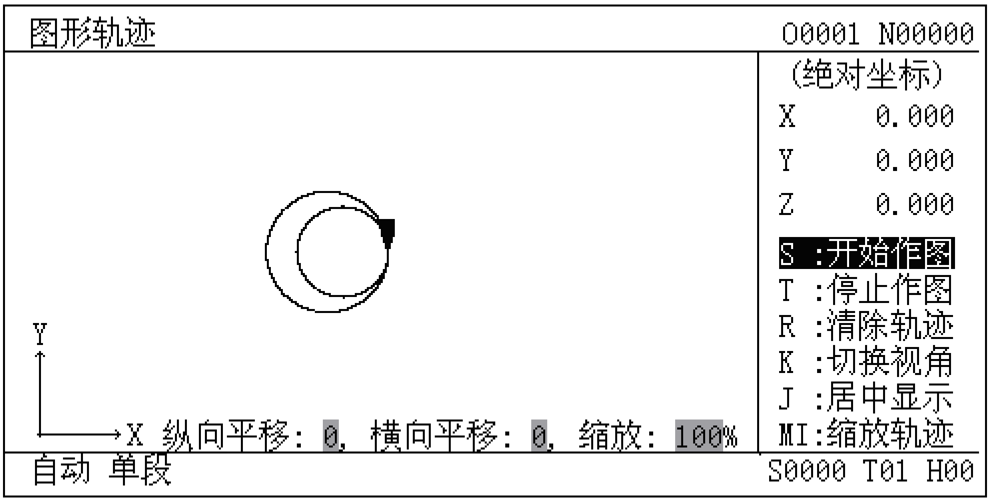
Scaling: Display the scaling of current graphic path.

Graphic center: Display the center of each axis.

Maximum, minimum: Set the maximum and minimum scope can be displayed by each axis.

C： Graphic track operation

Graphic track is as follows:



Vertical move: Display upper and lower part of the graphic.

Horizontal move: Display right and left part of the graphic.

Scaling: Display scaling of current graphic.

Absolute coordinate: Display the absolute coordinate of the program.

S：Start drawing, S is highlighted by pressing S key. Display drawing track.

T：Stop drawing, T is highlighted by pressing S key. I t stops drawing.

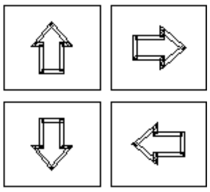
R：Clear graphic track, clear graphic track displayed before.

K：Switch view angle, coordinate value can be switched between 0~7 by pressing K key each time.

J： Display graphic in the center, that is, vertical move and horizontal move are 0.

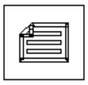
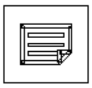
I：Scale up the track, the graphic is scaled up 2 fold by pressing I key once.

M：Scale down the track, the graphic is scaled down 2 fold by pressing M key once.

：Graphic moving up, down, left ,right.

9.1.3 Parameter setting

By the parameter setting, the characteristics of the drive unit and machine can be adjusted. See Appendix 1 for their significance

Press  key to enter the Parameter interface, then press  or  key to switch the parameter page, the figure is as follows:

```
BIT PARAMETER                                        O0000 N00000
   NO.      DATA        NO.      DATA        NO.      DATA
  _001   00000000       009   00011111       017   00101000
   002   00000010       010   00011111       018   00000000
   003   00000000       011   00000000       019   10000000
   004   01000000       012   00010011       020   00000000
   005   00010001       013   10000011       021   00000000
   006   00000000       014   00011111       022   00000000
   007   00000000       015   10000000       023   00000000
   008   00011111       016   00000000       024   00000000
*** *** ***  ACS  HWL  ***  *** ***
bit4:1/0:Analog vol./switch ctrl spindle
NO.  001
AUTO    SBK                                       S0000 T00 H00
```

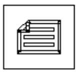
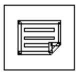
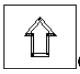
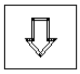
A Alteration of the bit parameter

1 Byte alteration

1） Turn on the parameter switch

2） Enter the MDI mode

3） Move the cursor to the parameter No. to be set

Method 1: Press  or  key to enter the page containing the parameter to be set, press or  key to move the cursor to the No. of the parameter to be set;

Method 2: Press address key , key in parameter No, then press  key.

4） Key in the new parameter value

5） Press key, the parameter value is entered and displayed

6） For security , the PARM SWT needs to be set to OFF after all parameters setting is finished

Example:

Set the BIT5 (DECI) of the bit parameter No.004 to 1, and the other bits unchanged.

Move the cursor to No.004, key in 01100000 by sequence in the prompt line, the figure is as follows:

```
BIT PARAMETER                                        O0000 N00000
   NO.        DATA       NO.        DATA       NO.        DATA
   001    00000000       009    00011111       017    00101000
   002    00000010       010    00011111       018    00000000
   003    00000000       011    00000000       019    10000000
  _004    00100000       012    00010011       020    00000000
   005    00010001       013    10000011       021    00000000
   006    00000000       014    00011111       022    00000000
   007    00000000       015    10000000       023    00000000
   008    00011111       016    00000000       024    00000000
*** RDRN DECI *** PROD *** *** SCW
bit5:1/0:DEC signal is low/high level
NO.   004 =  01100000_
MDI                                               S0000 T00 H00
```

Press DATA INPUT key to finish the parameter alteration. The page is as follows:

```
BIT PARAMETER                                        O0000 N00000
   NO.        DATA       NO.        DATA       NO.        DATA
   001    00000000       009    00011111       017    00101000
   002    00000010       010    00011111       018    00000000
   003    00000000       011    00000000       019    10000000
  _004    01100000       012    00010011       020    00000000
   005    00010001       013    10000011       021    00000000
   006    00000000       014    00011111       022    00000000
   007    00000000       015    10000000       023    00000000
   008    00011111       016    00000000       024    00000000
*** RDRN DECI *** PROD *** *** SCW
bit5:1/0:DEC signal is low/high level
NO.   004 =
MDI                                               S0000 T00 H00
```

2　Bit alteration

1）　Turn on the parameter switch

2）　Enter the MDI mode

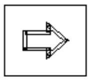
3）　Move the cursor to the No. of the parameter to be set

Method 1: Press or key to enter the page of the parameter to be set, press or key to move the cursor to the No. of the parameter to be set

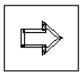
Method 2: Press address key key in parameter No., then press DATA INPUT key

4）　Press and hold key for 2 seconds or press CHANGE key to skip to a bit of the parameter, and the bit is backlighted. Press or key to move the cursor to the bit to be altered, then key in 0 or 1

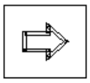
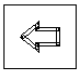
5）　After all parameters setting is finished, the PARM SWT needs to be set for OFF for security

Note: After entering a bit of the parameter, press and hold  key for 2 seconds or press  key, it may skip out of the bit and back to the parameter No.

Example:

Set the BIT5 (DECI) of the bit parameter No.004 to 1, and the other bits unchanged Move the cursor to "No.004" by the steps above, press and hold  key for 2 seconds or press  key to skip to a bit of the parameter, the figure is as follows:

```
BIT PARAMETER                                    O0000 N00000
   NO.      DATA       NO.      DATA       NO.      DATA
   001    00000000     009    00011111     017    00101000
   002    00000010     010    00011111     018    00000000
   003    00000000     011    00000000     019    10000000
  _004    01100000     012    00010011     020    00000000
   005    00010001     013    10000011     021    00000000
   006    00000000     014    00011111     022    00000000
   007    00000000     015    10000000     023    00000000
   008    00011111     016    00000000     024    00000000
*** RDRN DECI *** PROD *** *** SCW
bit7:1/0:Unused
NO.  004
MDI                                           S0000 T00 H00
```

Move the cursor to "BIT5" by pressing  or  key, the figure is as follows:

```
BIT PARAMETER                                    O0000 N00000
   NO.      DATA       NO.      DATA       NO.      DATA
   001    00000000     009    00011111     017    00101000
   002    00000010     010    00011111     018    00000000
   003    00000000     011    00000000     019    10000000
  _004    01000000     012    00010011     020    00000000
   005    00010001     013    10000011     021    00000000
   006    00000000     014    00011111     022    00000000
   007    00000000     015    10000000     023    00000000
   008    00011111     016    00000000     024    00000000
*** RDRN DECI *** PROD *** *** SCW
bit5:1/0:DEC signal is low/high level
NO.  004
MDI                                           S0000 T00 H00
```

Key in "1" to finish the alteration

BIT PARAMETER O0000 N00000

NO.	DATA	NO.	DATA	NO.	DATA
001	00000000	009	00011111	017	00101000
002	00000010	010	00011111	018	00000000
003	00000000	011	00000000	019	10000000
004	01100000	012	00010011	020	00000000
005	00010001	013	10000011	021	00000000
006	00000000	014	00011111	022	00000000
007	00000000	015	10000000	023	00000000
008	00011111	016	00000000	024	00000000

*** RDRN DECI *** PROD *** *** SCW
bit5:1/0:DEC signal is low/high level
NO. 004
MDI S0000 T00 H00

B Alteration of the data parameter, pitch data

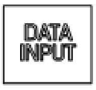
1 Data parameter alteration

1） Turn on the parameter switch;

2） Enter the MDI mode

3） Move the cursor to the No. of the parameter to be set

4） Key in the new parameter value

5） Press 

key, the value is entered and displayed

6） After all parameters setting is finished, the PARM SWT needs to be set to OFF for security

Example 1: Set the data parameter №059 to 4000.

Move the cursor to "№059" by the steps above, key in "4000" by sequence in the prompt line, the figure is as follows:

DATA PARAMETER O0000 N00000

NO.	DATA	NO.	DATA	NO.	DATA
049	1	057	1	065	100
050	1	058	1	066	100
051	1	059	7600	067	100
052	1	060	7600	068	100
053	1	061	7600	069	400
054	1	062	7600	070	8000
055	1	063	7600	071	50
056	1	064	100	072	100

Max. speed of rapid locating in X(mm/min)
NO. 059 4000_
MDI S0000 T00 H00

Press 

key to finish the alteration. The page is as follows

DATA PARAMETER O0000 N00000

NO.	DATA	NO.	DATA	NO.	DATA
049	1	057	1	065	100
050	1	058	1	066	100
051	1	_059	4000	067	100
052	1	060	7600	068	100
053	1	061	7600	069	400
054	1	062	7600	070	8000
055	1	063	7600	071	50
056	1	064	100	072	100

Max. speed of rapid locating in X(mm/min)
NO. 059
MDI S0000 T00 H00

Example 2: Set the X axis value of the pitch data No.000 to 12, set the value of Z axis to 30 Move the cursor to pitch data No.000 by the steps above, key in “X12” by sequence in the cue line, the figure is as follows:

SCREW-PITCH PARAMETER O0000 N00000

NO.	X	Y	Z	C	NO.	X	Y	Z	C
_000	0	0	0	0	008	0	0	0	0
001	0	0	0	0	009	0	0	0	0
002	0	0	0	0	010	0	0	0	0
003	0	0	0	0	011	0	0	0	0
004	0	0	0	0	012	0	0	0	0
005	0	0	0	0	013	0	0	0	0
006	0	0	0	0	014	0	0	0	0
007	0	0	0	0	015	0	0	0	0

UNIT:0.001(mm)
NO. 000 X 12_
MDI S0000 T00 H00

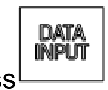
Pres 

key to finish the alteration. The page is as follows:

SCREW-PITCH PARAMETER O0000 N00000

NO.	X	Y	Z	C	NO.	X	Y	Z	C
_000	12	0	0	0	008	0	0	0	0
001	0	0	0	0	009	0	0	0	0
002	0	0	0	0	010	0	0	0	0
003	0	0	0	0	011	0	0	0	0
004	0	0	0	0	012	0	0	0	0
005	0	0	0	0	013	0	0	0	0
006	0	0	0	0	014	0	0	0	0
007	0	0	0	0	015	0	0	0	0

UNIT:0.001(mm)
NO. 000
MDI S0000 T00 H00

The same as above, key in “Z30”by sequence in the prompt line, press 

key to finish the alteration. The page is as follows:

SCREW-PITCH PARAMETER O0000 N00000

NO.	X	Y	Z	C	NO.	X	Y	Z	C
000	12	0	30	0	008	0	0	0	0
001	0	0	0	0	009	0	0	0	0
002	0	0	0	0	010	0	0	0	0
003	0	0	0	0	011	0	0	0	0
004	0	0	0	0	012	0	0	0	0
005	0	0	0	0	013	0	0	0	0
006	0	0	0	0	014	0	0	0	0
007	0	0	0	0	015	0	0	0	0

UNIT:0.001(mm)
NO. 000
MDI S0000 T00 H00

9.2 The Password Setting and Alteration

To prevent the part programs, CNC parameters from malignant alteration, this GSK980MD provides an authority setting function that is graded for 4 levels. By decending sequence, they are machine builder (2nd) level, equipment management (3rd) level, technician (4th) level, machining operation (5th) level

The 2nd level: Modification of the CNC bit parameter, data parameter, pitch data, tool offset data, part program edit, PLC ladder transmission etc. are allowed

The 3rd level: initial password 2345, the CNC bit parameter, data parameter, tool offset data, part program edit operations are allowed;

The 4th level: initial password 1234, tool offset data (for tool setting), macro variables, part program edit operations are allowed; but the CNC bit parameter, data parameter, pitch data operations are unallowed.

The 5th level: no password. Only the machine panel operation is allowed, and the operations of part program edit and selection, the alteration operations of CNC bit parameter, data parameter, pitch data, tool offset data are unallowed

AUTH. OPERATION O0000 N00000

CURRENT LEVEL: 3	Backup PAR. (User))
SET LOWER LEVEL	Resume PAR. (User))
► INPUT PASSWORD:________	Resume PAR.1 (Test)
UPDATE PASS. :________	Resume PAR.2 (Step)
	Resume PAR.3 (Servo)

Modify parameter and edit program
MDI S0000 T00 H00

After entering the authority setting page, the cursor locates at the "INPUT PASSWORD:"line. It

may press the ⇧ or ⇩ key to move the cursor to the corresponding item.

- Press ⇧ key once, the cursor shifts a line upward. If the current cursor locates at the "SET LOWER LEVEL"line (1st line) , press ⇧ key, the cursor shifts to the "UPDATE PASS:"line (end line)
- Press ⇩ key once, the cursor shifts a line upward. If the current cursor locates at the end line, by pressing ⇩ key once, the cursor moves to the 1st line.

9.2.1 Entry of the operation level

1 After entering the PASSWORD SETTING page, move the cursor to the "INPUT PASSWORD:"line;

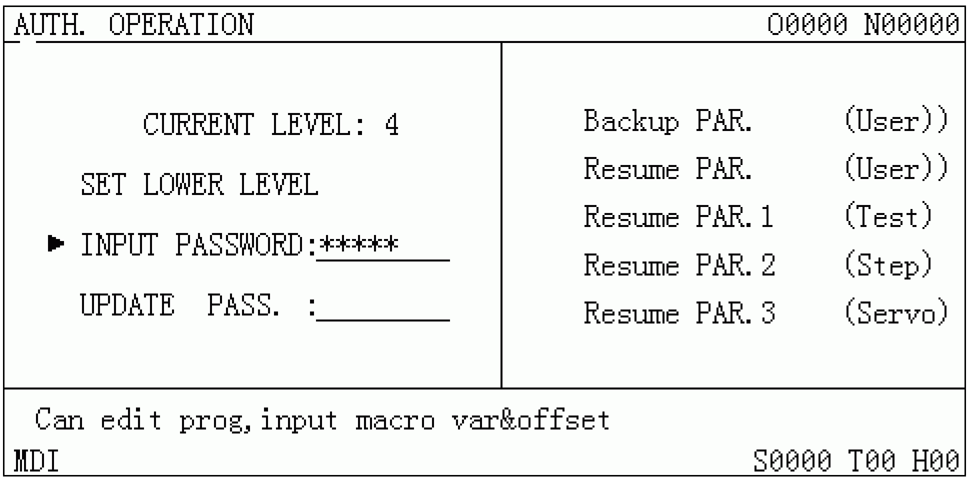
2 Key in the password (an "*"sign added each time inputting a character)

3 Press DATA INPUT key to finish the inputting, and it will enter the corresponding password level.

Note The length of this GSK980MD system password corresponds to the operation level, which can't be added or decreased by user at will.

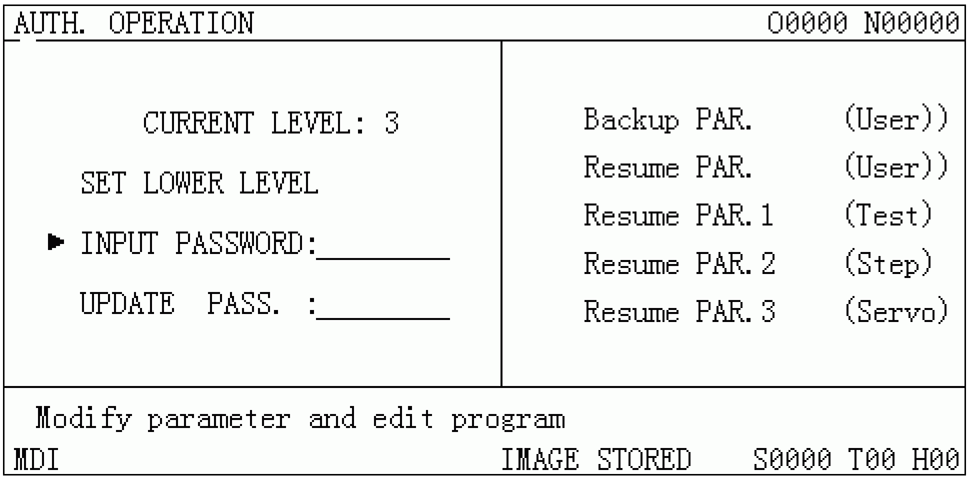
Operation level	Password length	Initial password
3rd	5 bits	12345
4th	4 bits	1234
5th	No	No

Example: The current CNC level is the 4th level, as the following page shows. The 3rd level password of CNC is 12345, please alter the current level to the 3rd level.



Move the cursor to the “INPUT PASSWORD:”line, key in 12345, then press the 

key, the CNC prompts “Modify parameter and edit program”, “Password passed”, and the current level is the 3rd level. The page is as follows:



Note: When current operation authority is lower than or equal to the 3rd level (3rd, 4th, 5th level), the password level is not changed if repower the CNC system. If previous level is higher than the 3rd level (0, 1st, or 2nd level), it defaults the 3rd level.

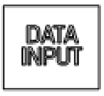
Volume II Operation

9.2.2 Alteration of the password

Steps for password alteration:

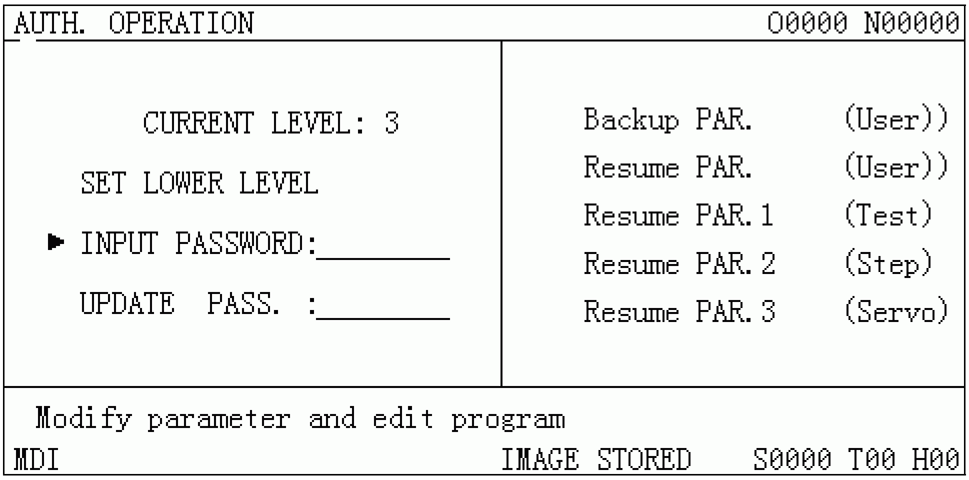
1 After entering the PASSWORD SETTING page, enter the password by the methods in Section10.3.2;

2 Move the cursor to the“ALTER PASSWORD:”line;

3 Key in the new password, and press 

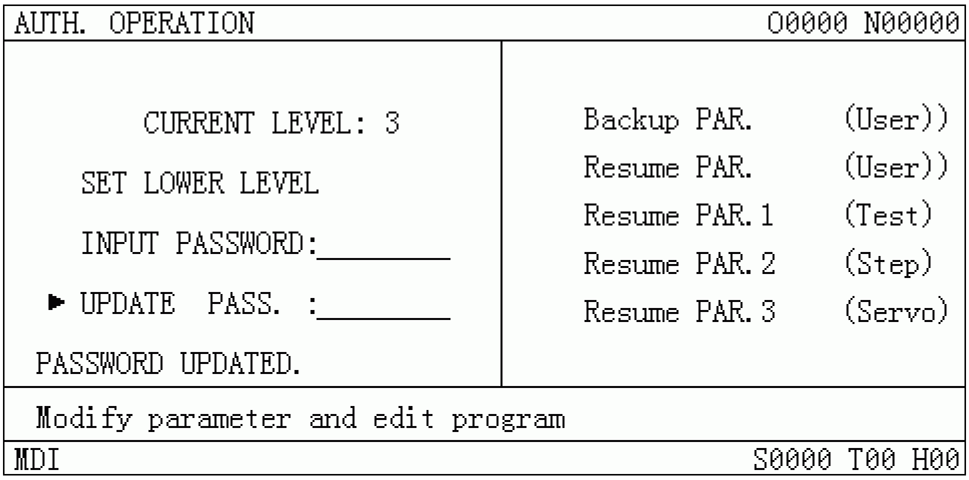
key

4 The CNC system prompts “PLEASE INPUT USER PASSWORD AGAIN”, the page is as follows:

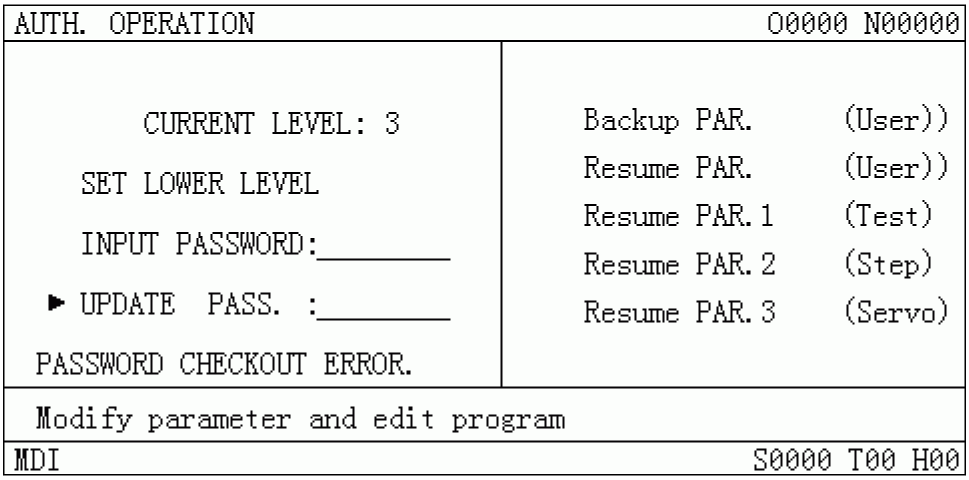


5 After reinputting the password, press 

key, if the two passwords input are identical, CNC prompts “PASSWORD UPDATED”. So the password alteration is successful.



6 If the two passwords input are not identical, CNC prompts “PASSWORD CHECKOUT ERROR.”, the page is as follows:



9.2.3 Lower level set

The demotion of the operation level is used to enter a lower level from a higher level, the steps are as follows:

1 After entering the PASSWORD SETTING page, key in the password by the method in Section 10.3.2

2 Move the cursor to the“SET LOWER LEVEL”line, if the current CNC operation is the 3rd level, the page is as follows:

```
AUTH. OPERATION                              O0000 N00000
---------------------------------------------------------
       CURRENT LEVEL: 3         |  Backup PAR.     (User))
                                |  Resume PAR.     (User))
  ► SET LOWER LEVEL             |  Resume PAR.1    (Test)
    INPUT PASSWORD:________     |  Resume PAR.2    (Step)
    UPDATE  PASS.  :________    |  Resume PAR.3    (Servo)
---------------------------------------------------------
 Modify parameter and edit program
MDI                                        S0000 T00 H00
```

3 Press 

key, the CNC prompts "CURRENT LEVEL TO 4, OK？ "; the page is as follows:

```
AUTH. OPERATION                              O0000 N00000
---------------------------------------------------------
       CURRENT LEVEL: 3         |  Backup PAR.     (User))
                                |  Resume PAR.     (User))
  ► SET LOWER LEVEL             |  Resume PAR.1    (Test)
    INPUT PASSWORD:________     |  Resume PAR.2    (Step)
    UPDATE  PASS.  :________    |  Resume PAR.3    (Servo)
  CURRENT LEVEL TO4,MAKE SURE?  |
---------------------------------------------------------
 Modify parameter and edit program
MDI                                        S0000 T00 H00
```

4 Press 

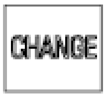
key again, if the demotion is successful, the page is as follows:

```
AUTH. OPERATION                              O0000 N00000
---------------------------------------------------------
       CURRENT LEVEL: 4         |  Backup PAR.     (User))
                                |  Resume PAR.     (User))
  ► SET LOWER LEVEL             |  Resume PAR.1    (Test)
    INPUT PASSWORD:________     |  Resume PAR.2    (Step)
    UPDATE  PASS.  :________    |  Resume PAR.3    (Servo)
---------------------------------------------------------
 Can edit prog,input macro var&offset
MDI                                        S0000 T00 H00
```

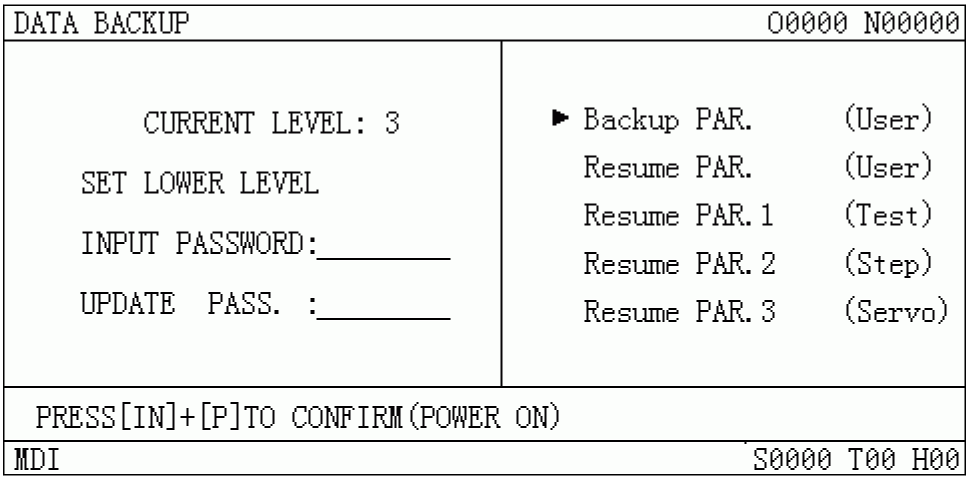
Note If the current level is the 5th level, the demotion operation is unallowed.

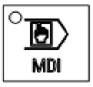
9.3 Data Restore and Backup

The user data (such as bit parameter and pitch data) can be backup (saved) and restored (read) in this GSK980MD system. It doesn't affect the part programs stored in the CNC system while backuping and restoring these data. The backup page is as follows:

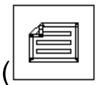
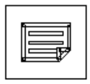
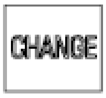
Press 

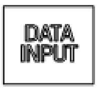
key repeatedly, "PASSWORD SETTING" and "DATA BACKUP" pages can be switched.



- Turn on the parameter switch
- Press 

key to enter the MDI mode, then press 

key ( or  key if necessary) to enter PASSWORD SETTING page;
- Press 

, and switch to the Data Backup page.
- Move the cursor to the desired item;
- Press 

. 

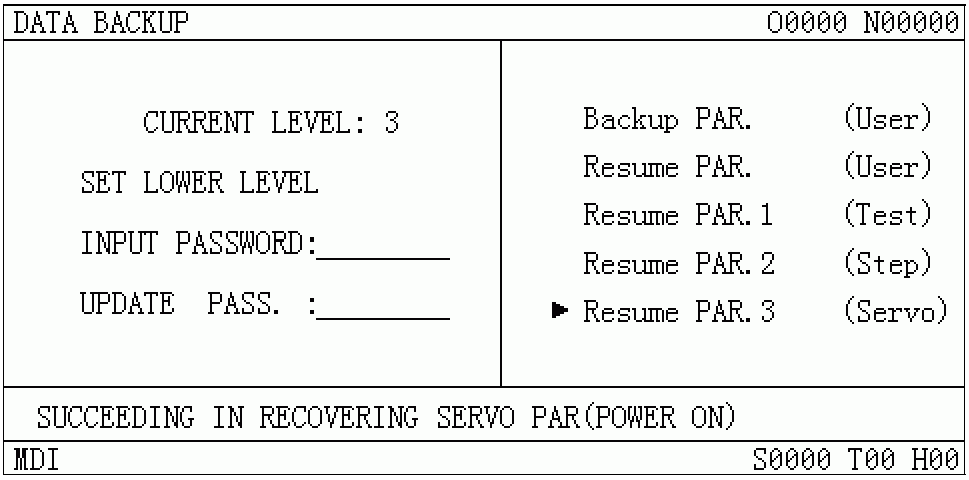
keys together.

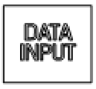
Note Don't cut off the power in the backup and restore operation of the data, and no other operation is suggested to be performed before the aforesaid operation is prompted to be finished.

Example: to restore the CNC parameter to 1μ level servo standard parameter, the steps are as follows:

Turn on the parameter switch, and enter the Backup PAR. page of MDI mode, move the cursor to "Recover Default PAR. (1μ level)", as the following figure shows:

Volume II Operation

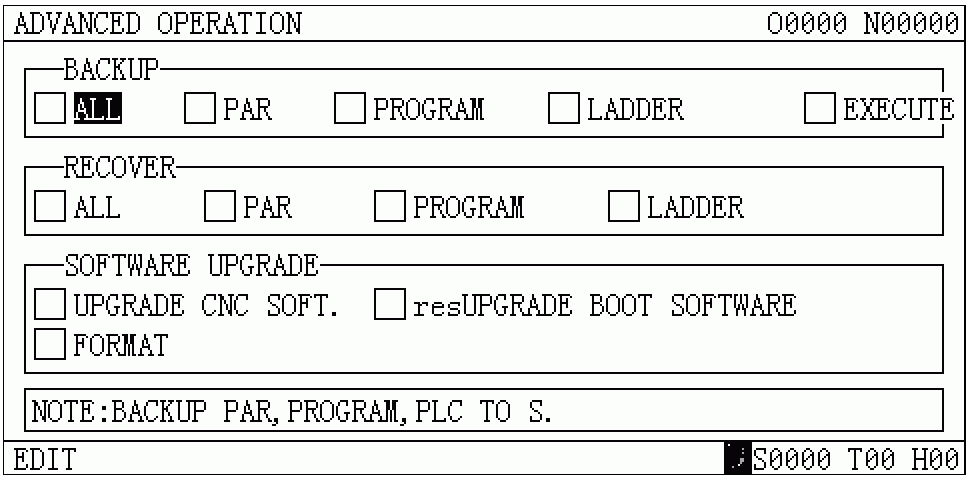


Press 

keys together, the CNC system prompts “SERVO PAR BACKUP RECOVERED (POWER ON)”.

CHAPTER 10 ADVANCE OPERATION

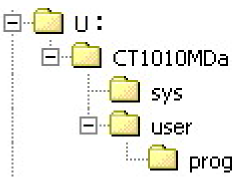
Advance operation interface of GSK980MDa, which is as follows, is started by connecting CNC to USB. In this interface, communication between CNC & USB and system update operations can be done. Its transmission speed is much faster than traditional serial communication speed, greatly increases the efficiency of file transmission. More over, USB is easy to carry, to use and it supports hot plugging, plug and play at once.



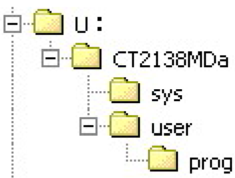
10.1 Operation path

USB operation in 980MDa is searching and setting up destination list on U disk with its number. Therefore, the system with different number is corresponding to different U disk list in advance operation.

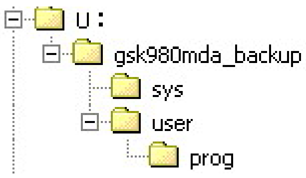
Example: If the number of system A is CT1010MDa, the list of advance operation on U disk is as follows:



If the number of system B is CT2138MDa, the list of advance operation on U disk is as follows:



If the system has no number, the list of advance operation on U disk is as follows:



Note: The number of the system can be found in version information page of diagnosis. The following contents are described by list of gsk980mda_backup.

➢ **Path explanations**

Path file folder		Explanation
user\		Target position for parameter and PLC file backup and restore
	prog\	Target position for part program file backup and restore

➢ **File specification**

	File name	Expended name	Remark
Parameter file	Para1, Para2, Para3	.par	Case sensitive
Part program	O0000 ~ O9999	.CNC	Case sensitive
PLC file	plc ~ plc7	.ldx	Case sensitive

➢ **Operation authority**

Backup operation	Parameter	Authority level 3 (including level 3)
	Part program	Authority level 3 (including level 3)
	Ladder diagram	Authority level 3 (including level 3)
Restore operation	Parameter	Authority level 3 (including level 3)
	Part program	Authority level 3 (including level 3)
	Ladder diagram	Authority level 2 (including level 2)

Note: Level 2 or above authority is needed for part program operation above number 9000.

10.2 Operation instructions

- **Key descriptions**

Cursor moving：Press direction keys ⇧ ⇩ ⇨ ⇦ CHANGE to move the cursor.

Menu selection: Press DATA INPUT key to select the operation item which cursor is in.

Menu cancellation: Press CANCEL key to cancel the operation item which cursor is in.

Operation execution：Press DATA OUTPUT key to execute all operation items selected in current column.

Operation confirmation：Execution needs to be confirmed, please press Y & key to confirm or press N key to cancel the execution.

- **Parameter restore and backup**

Backup the parameter: Copy all parameter states and values to U:\gsk980MDa_backup\user\ of USB memory unit in the form of file Para1.par，Para2.par，Para3.par. If the above-mentioned file does not exist, set up a new one: If the file exists, this file will be overwritten by the new one.

Restore the parameter: Copy parameter files from USB memory unit U:\gsk980MDa_backup\user\ back to the CNC system to restore the system parameter. Restore operation cannot be done if the above-mentioned path is moved or altered or irregular file name is renamed.

Note: Repower the CNC system after parameter load is successful.

- **Part program restore and backup**

Backup the part parameter: Copy all part programs of current system to U:\gsk980MDa_backup\user\prog\ of USB memory unit in the form of file .CNC. If the above-mentioned file does not exist, set up a new one: If the file exists, this file will be overwritten by the new one.

Restore the part program: Copy all part programs from USB memory unit U:\gsk980MDa_backup\user\prog\ back to the CNC system to restore the part program. Restore operation cannot be done if the above-mentioned path is moved or altered or irregular file name is renamed.

- **Ladder diagram (PLC) restore and backup**

The ladder diagram backup: Copy all ladder diagrams (.ldx file) of the current system to U:\gsk980MDa_backup\user\ of USB memory unit. If the above-mentioned file does not exist, set up a new one: If the file exists, this file will be overwritten by the new one.

Restore the ladder diagram: Copy parameter files from USB memory unit U:\gsk980MDa_backup\user\ back to the CNC system to restore the ladder diagram. Restore operation cannot be done if the above-mentioned path is moved or altered or irregular file name is renamed.

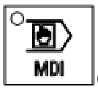
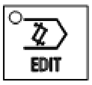
Note: Repower the CNC system after the ladder diagram restore is successful.

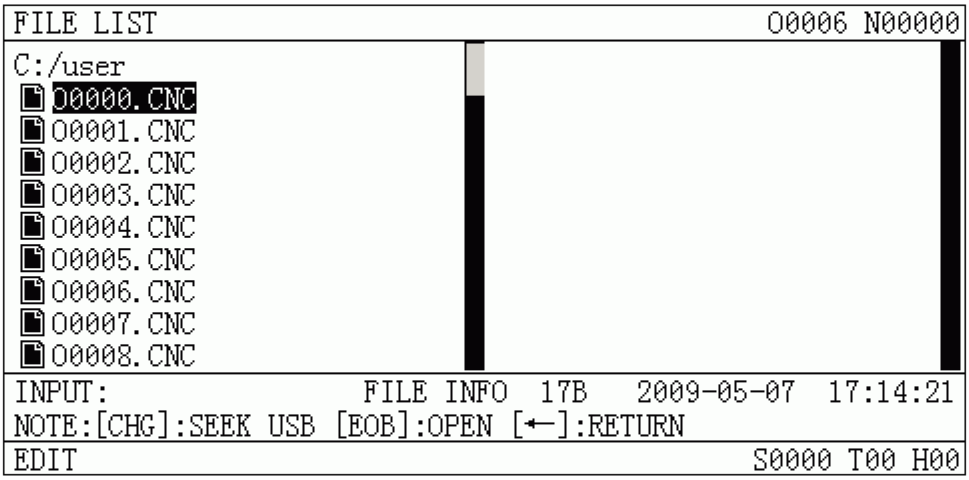
10.3 Attentions

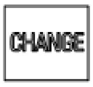
- **Notice：**If a file or list on target path has the same name as the one will be copied, it will be overwritten and replaced by the system automatically. Therefore, to prevent the file or list from overwriting or replacing, please copy and save it separately.
- It forbids doing any other operation in advance operation. Once operation is performed, it can not be interrupted until it is finished.
- If the file to be saved or restored is large, operation time will be long. Please wait.
- Pull out USB if abnormal conditions occur, then connect it again.

CHAPTER 11 FLASH OPERATION

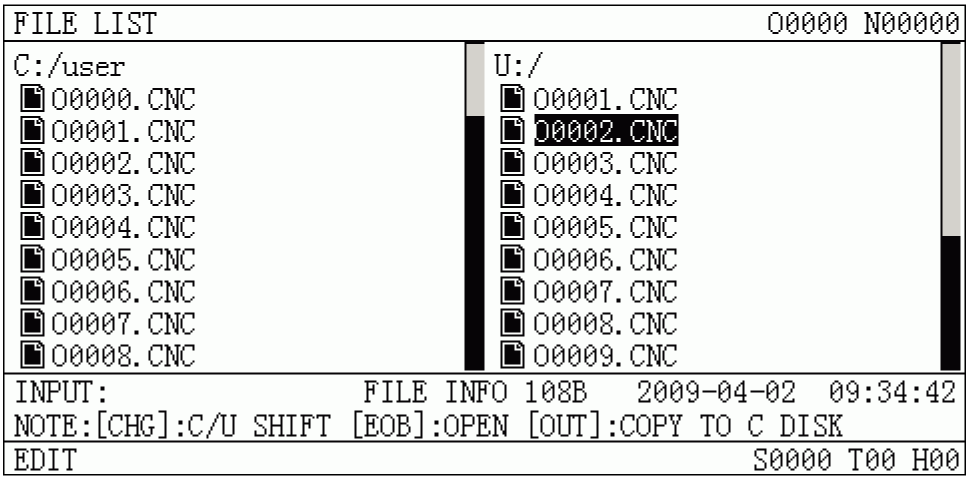
11.1. File list

Press or  key to select［MDI］or [EDIT] mode, press  key to enter［file list interface, the page is as follows：



In edit or MDI mode, press  key to identify U disk.

If identification is unsuccessful, it prompts: “Fail to connect U disk”. If identification is successful, the following file list will be displayed.



Special explanation:

The list information of disk CNC is displayed at the page left and list information of disk USB is displayed at the page right. The display column will not display any information if U disk is not detected. Character entry box, file attributes information and user operation prompts are displayed at the bottom of the page.

1. Current list page only display the list information of the currently opened folder.
2. U disk can be identified in edit or MDI mode.

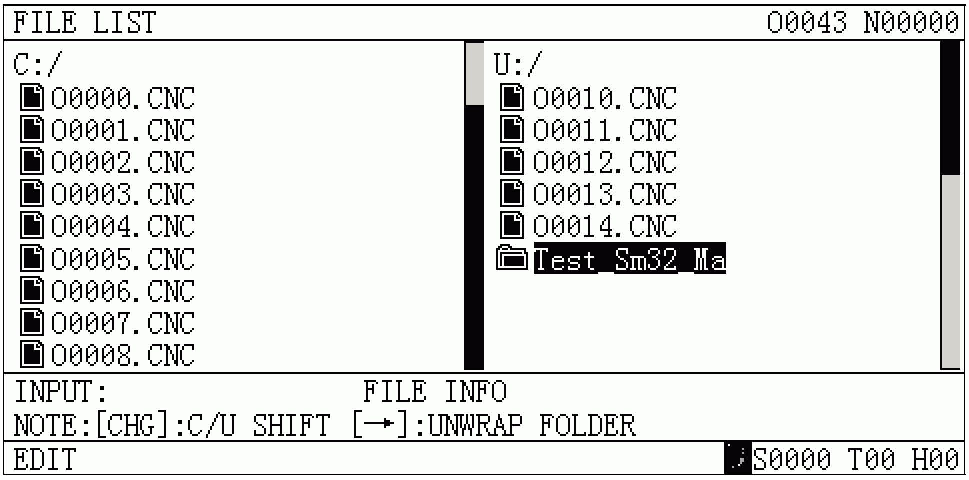
3. It not support Chinese complex characters.
4. It not support Chinese long file name, only the first three characters .+“～1”of this file name can be displayed.
5. Non-CNC file of C disk and U disk is displayed.

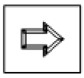
Note: The file name，which consists of “O”+“4 digits”+“.CNC ”, is considered to be CNC format file.

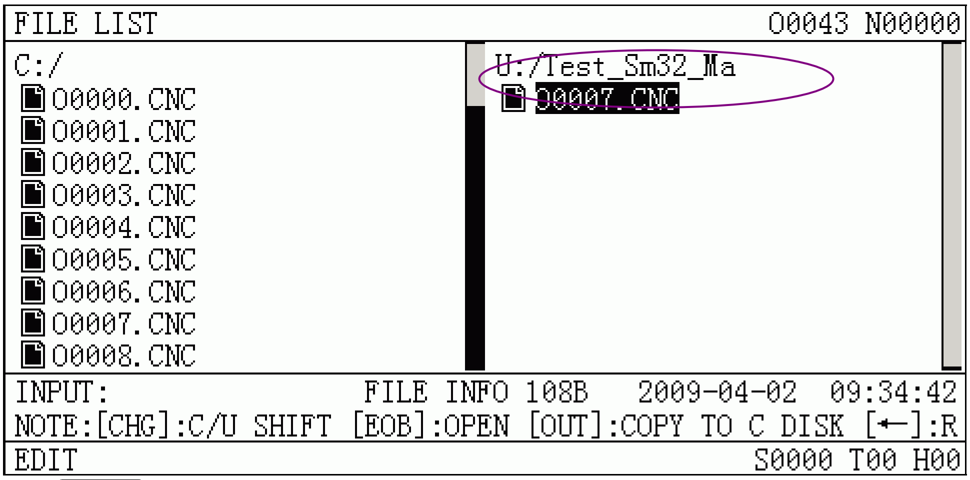
11.2. Introduction of general file operation function

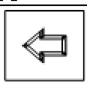
11.2.1 Open and close file folder

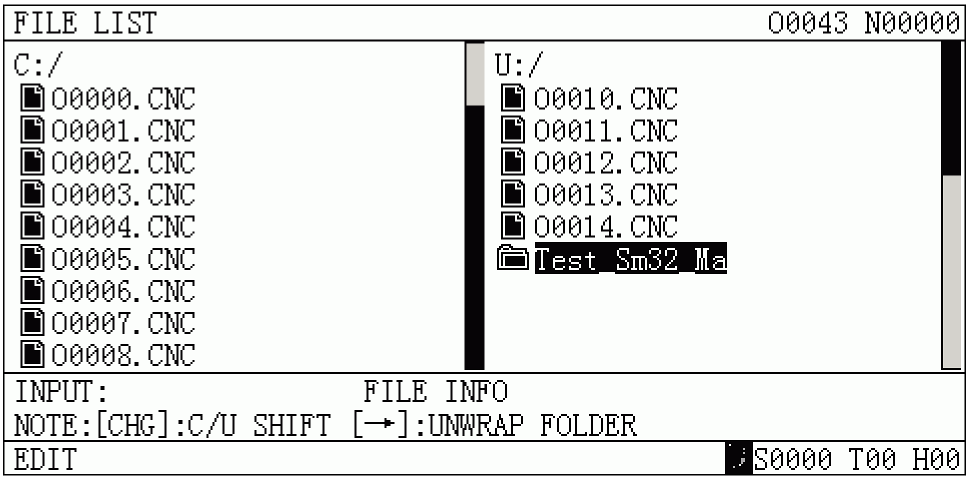
Move the cursor to the folder will be opened.



Press  key to open the folder. The list which the file locates is displayed in the first line (long list is scrolling display)



Press  key to close the folder and return to the next higher level of the list.



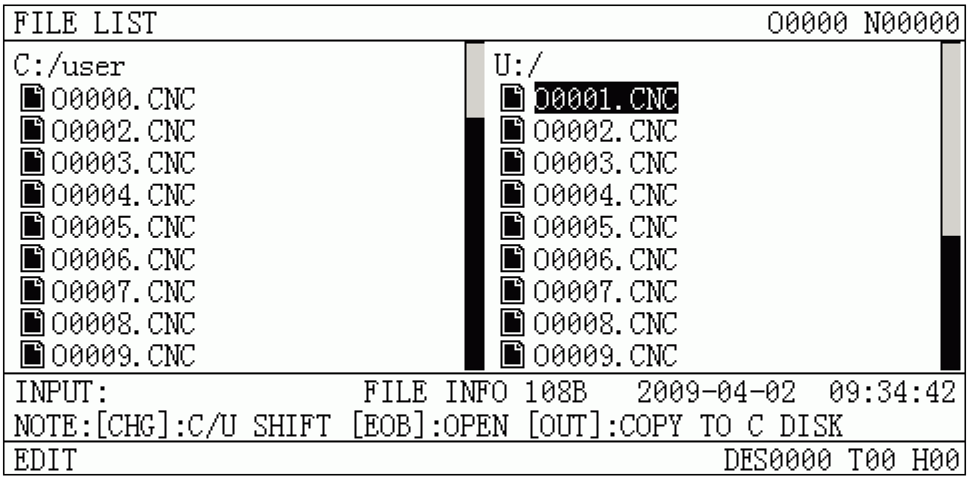
11.2.2 Copy the file by one key(current list in C disk←→current list in U disk)

In “edit”mode, select the CNC format file, press 

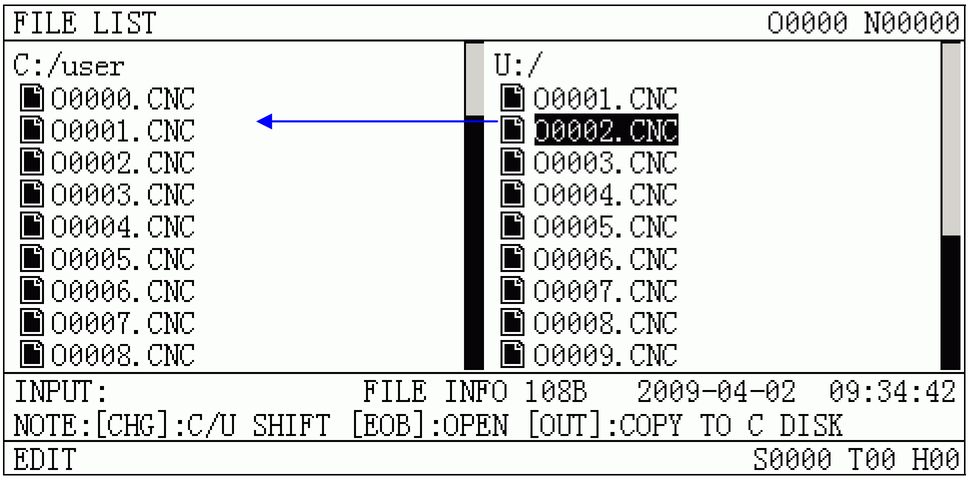
key to copy it. See the following figure：

① Select CNC file, press 

；

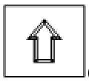
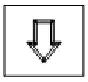


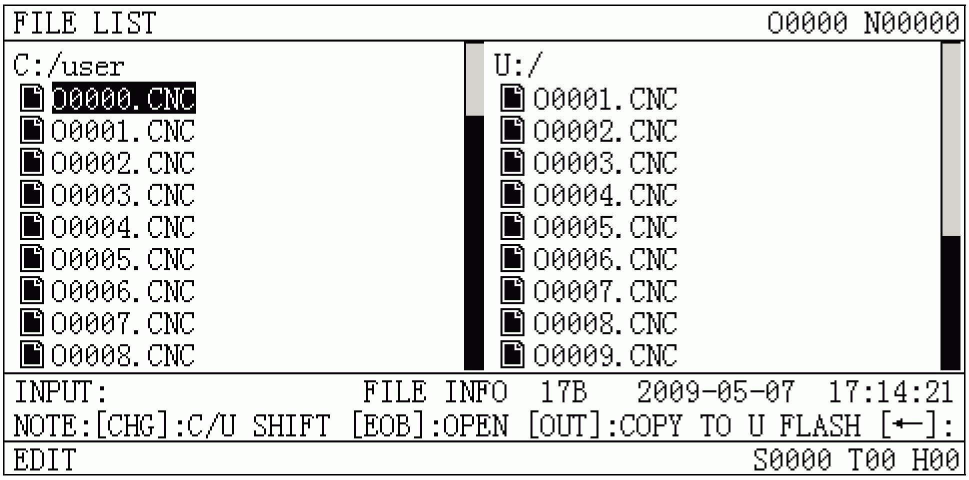
② After duplication is successful, the cursor moves to the next file in current list. The list on the other side is refreshed at once.



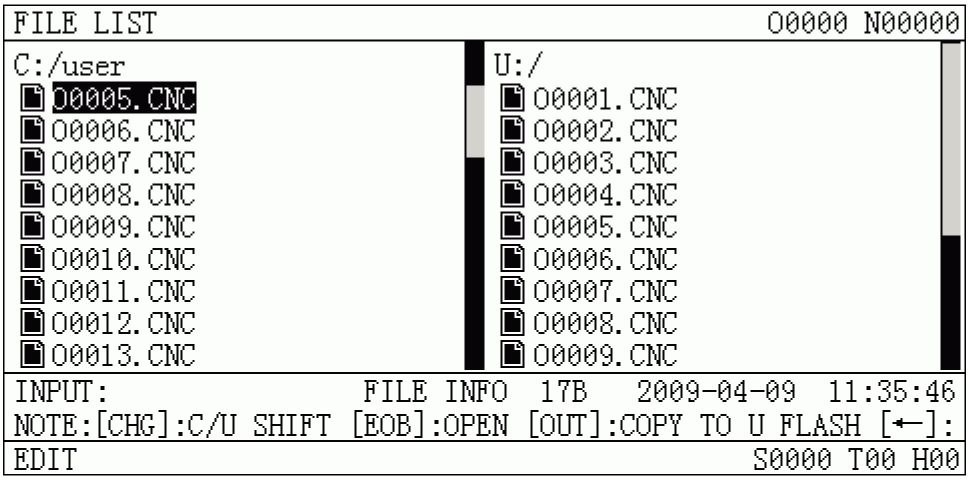
Special explanation：Duplication can not be done under 5-level authority.

11.2.3 CNC file search

In “EDIT”and“AUTO”mode, input target program number in input column, and press  or  to search this program.

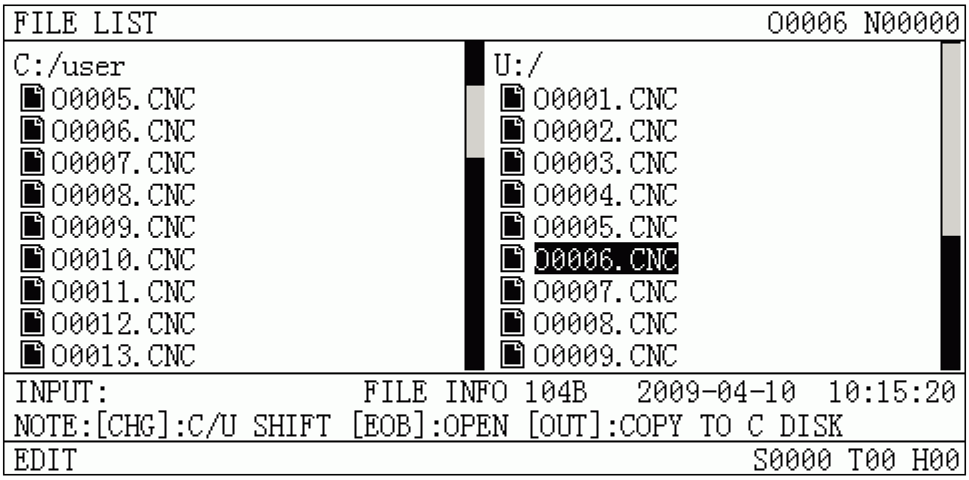


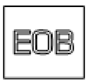
If program search is successful after input “O5”, the cursor moves to target program. If this program can not be searched, “the file dose not exist” will be prompted at message column.



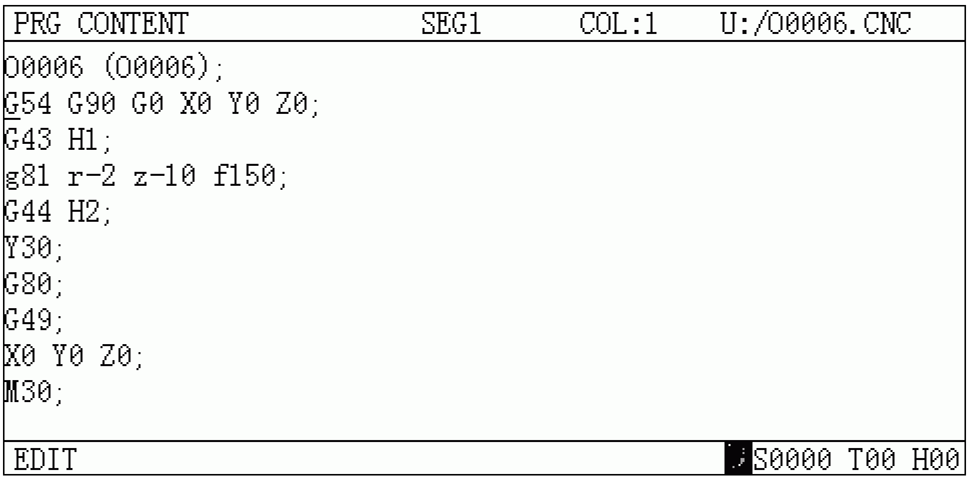
11.2.4 Open CNC file

1. In"EDIT"and"AUTO"mode, select the CNC format file when there is no program execution.



2. Press 

key to open the file. Current page is switched to［program content］page.



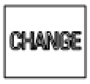
Special explanations:

1. The program above number 9000 can not be opened with authority level 3 or under

level 3.

2. The program file can not be opened with authority level 5.

Attentions:

1. In “program content”, it is not allowed to do any operation on U disk. These operations are: setting-up, duplication, rename, deletion, editing, save, etc.. Process and check operations can be done for programs on U disk in page“program content”.
2. The called subprogram in auto-run should in a same level of list with main program.
3. Pull out U disk when it is open, system alarm occurs“U disk is not connected”. At this time, plug in U disk again, press 

key to detect U disk in MDI mode, or press 

+ 

keys to clear the alarm.

VOLUME Ⅲ INSTALLATION

CHAPTER 1 INSTALLATION LAYOUT

1.1 GSK980MDa Connection

1.1.1 GSK980MDa back cover interface layout

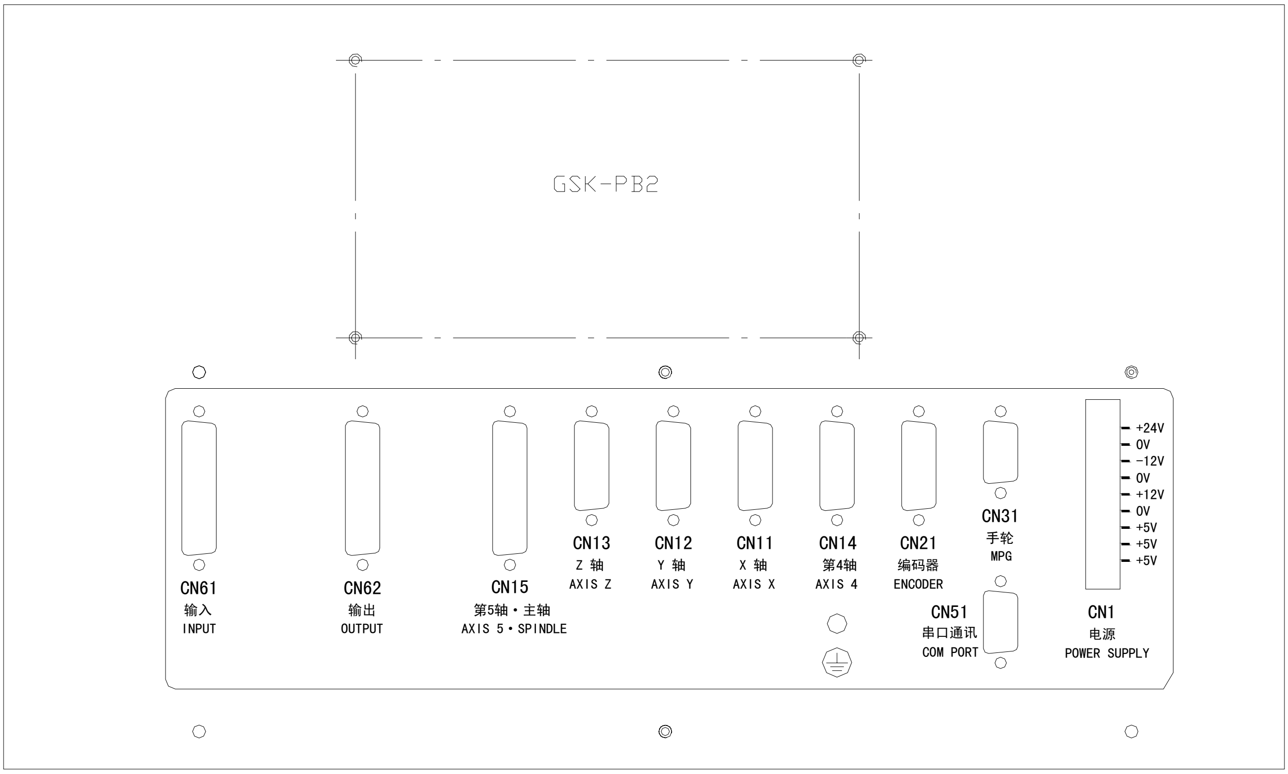


Fig 1-1 GSK980MDa back cover interface layout

1.1.2 Interface explanation

- Power box: GSK-PB2,for +5V, +24V, +12V, -12V, GND power supply
- CN11: X axis, 15-core DB female socket,for connecting X axis drive unit
- CN12: Y axis, 15-core DB female socket,for connecting Y axis drive unit
- CN13: Z axis, 15-core DB female socket,for connecting Z axis drive unit
- CN14: 4th axis，15-core DB female soket,for connecting 4th axis drive unit
- CN21: coder, 15-core DB female socket,for connecting Encoderd
- CN51: inverter, 9-core DB male socket,for connecting pc RS232 interface
- CN15: 5th axis&spindle port, 25-core DB male socket,for connecting inverter & 5th axis
- CN31: handwheel, 26-core 3 line famele socket,for connecting handwheel;
- CN62: ouput，44-core 3 lines famele socket，for sending the signal of CNC to machine
- CN61:input, 44-core 3 line male socket，for sending the signal of machine to CNC

1.2 GSK980MDa Installation

1.2.1 GSK980MDa external dimensions

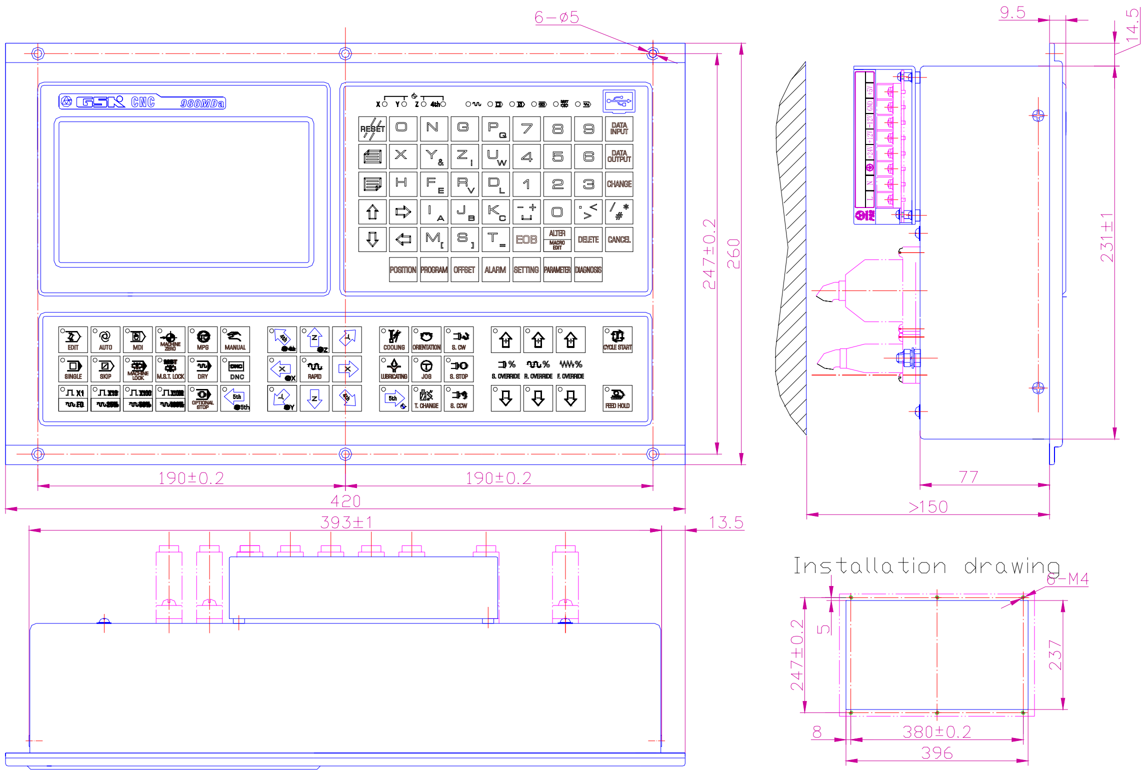


Fig. 1-2 GSK980MDa external dimensions

1.2.2 Installation conditions of the cabinet

- The dust, cooling liquid and organic resolution should be effectively prevented from entering the cabinet;
- The designed distance between the CNC back cover and the cabinet should be not less than 20cm, the inside and outside temperature difference of the cabinet should be no les than 10℃ temperature rises when the cabinet inside temperature rises;
- Fans should be fixed in the cabinet to ventilate it;
- The panel should be installed in a place where the coolant can't splash;
- The external electrical interference should be taken into consideration in cabinet design to prevent it from transferring to CNC system.

1.2.3 Protection methods against interference

In order to ensure the CNC stable working, the anti-interference technology such as space electromagnetic radiation shielding, impact current absorbing, power mixed wave filtering are employed in CNC design.And the following measures are necessary during CNC connection:

1. Make CNC far from the interference devices (inverter, AC contactor, static generator, high-pressure generator and powered sectional devices etc.);

2. To supply the CNC via an isolation transformer , the machine with the CNC

should be grounded, the CNC and drive unit should be connected with independent grounding wires at the grounding point;

3. To supress interference: connect parallel RC circuit at both ends of AC coil (Fig. 1-4), RC circuit should approach to inductive loading as close as possible; reversely connect parallel freewheeling diode at both ends of DC coil (Fig. 1-5); connect parallel surge absorber at the ends of AC motor coil (Fig. 1-6);

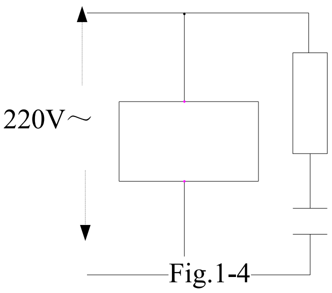


Fig.1-4

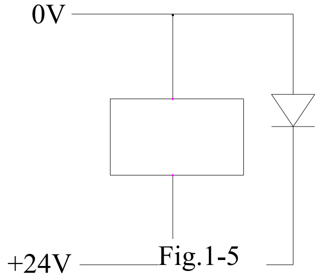


Fig.1-5

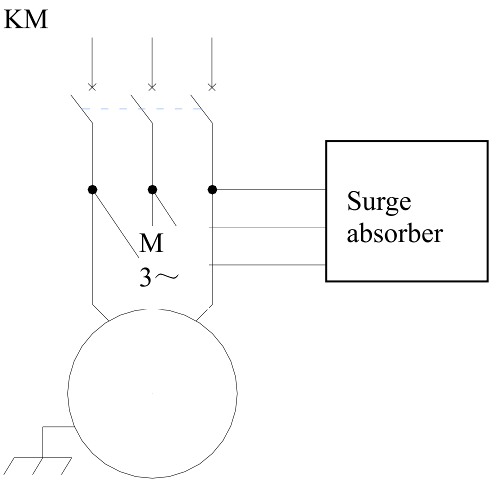


Fig.1-6

4. To employ with twisted shield cable or shield cable for the leadout cable of CNC, the cable shield tier is grounded by single end at CNC side, signal cable should be as short as possible;

5. In order to decrease the mutual interference between CNC cables or CNC cables with strong-power cables,the wiring should comply to the following principles:

Group	Cable type	Wiring requirement
A	AC power line	Tie up A group cables with a clearance at least 10cm from that of B, C groups, or shield A group cables from electromagnetism
	AC coil	
	AC contactor	
B	DC coil（24VDC）	Tie up B and A group cables separately or shield B group cables; and the further B group cables are from that of C group, the better it is
	DC relay（24VDC）	
	Cables between CNC and strong-power cabinet	
	Cables between CNC and machine	
C	Cables between CNC and servo drive unit	Tie up C and A group cables separately, or shield C group cables; and the cable distance between C group and B group is at least 10cm with twisted pair cable applied.
	Position feedback cable	
	Position encoder cable	
	MPG cable	
	Other cables for shield	

CHAPTER 2 DEFINITION&CONNECTION OF INTERFACE SIGNALS

2.1 Connection to Drive unit

2.1.1 Drive interface definition

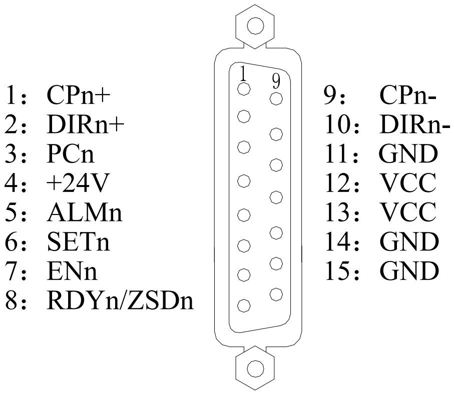


Fig.2-1 CN11, CN12, CN13 interface（DB15 female）

Signal	Explanation
CPn+, CPn-	Command pulse signal
DIRn+, DIRn-	Command direction sigal
PCn	Zero signal
ALMn	Drive unit alarm signal
ENn	Axis enable signal
SETn	Pusle disable signal

2.1.2 Command pulse and direction signals

nCP+，nCP- are command pulse signals, nDIR+，nDIR- are command direction signals. These two group signals are both difference output（AM26LS31）, the interior circuit for them is shown in Fig. 2-2.

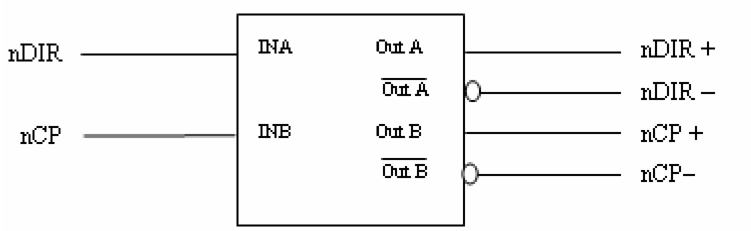


Fig. 2-2 Interior circuit of command pulse and direction signals

2.1.3 Drive unit alarm signal

The low or high level of the drive unit alarm level is set by the CNC bit parameter No.009 BIT0～BIT4，whose interior circuit is shown in Fig. 2-3:

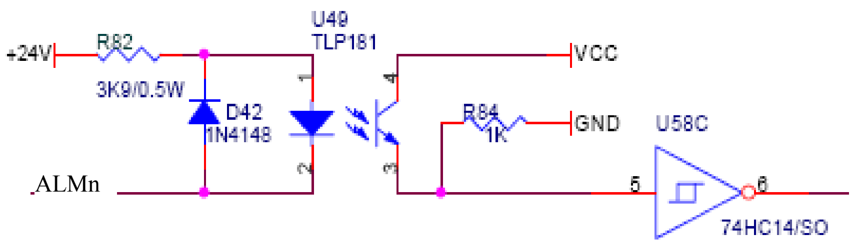


Fig.2-3 interior circuit of drive unit alarm signal

This input circuit requires that the drive unit transmits signal by the following types in Fig. 2-4:

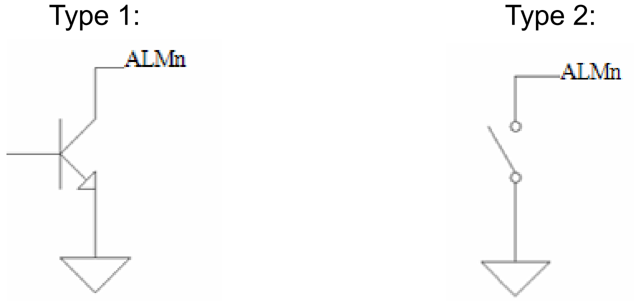


Fig.2-4 Signal types of drive unit

2.1.4 Axis enable signal ENn

nEN signal output is valid as CNC works normally (nEN signal to 0V); when the drive unit alarm or emergency alarm occurs, CNC cuts off nEN signal output (nEN signal to0V off). The interior interface circuit is shown in Fig.2-5:

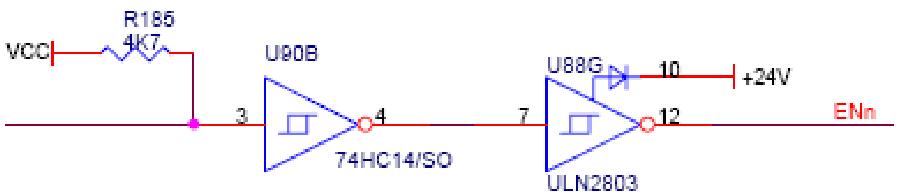


Fig.2-5 interior interface circuit for axis enable signal

2.1.5 Pulse disable signal SETn

nSET signal is used to control servo input disable which can enhance the anti-disturbance capability between CNC and drive unit. This signal is at low level if there is pulse output from CNC, high resistance if not. The interior interface circuit of it is shown in Fig. 2-6:

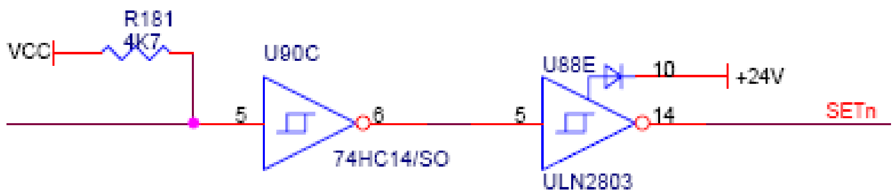


Fig.2-6 Interior interface circuit for pulse disable signal

2.1.6 Zero signal nPC

The one-rotation or approach switch signal is taken as zero signal for machine zero return. Its interior connection circuit is shown in Fig.2-7.

Volume Ⅲ Installation

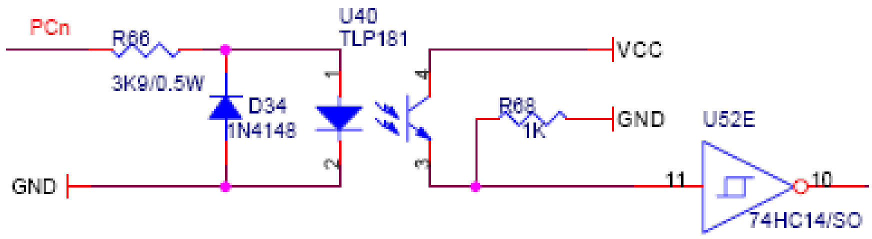


Fig.2-7 Zero signal circuit

Note: nPC signal uses +24V level.

a) The connection for NPN Hall elements taken as both deceleration signal and zero signal is shown in Fig. 2-8:

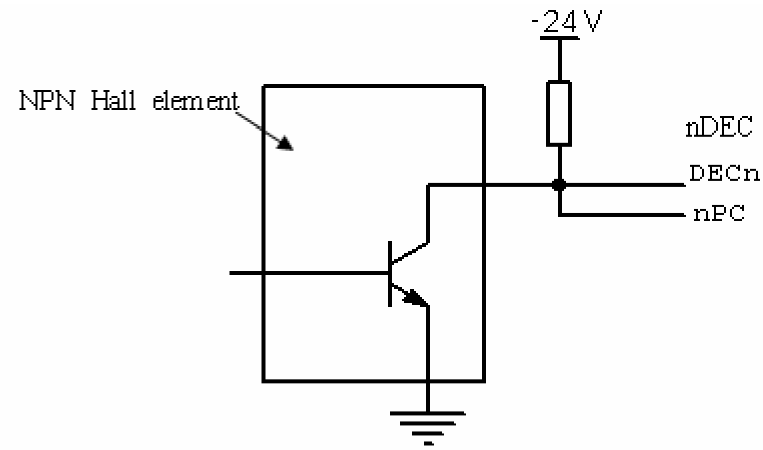


Fig. 2-8 Connection using NPN Hall elements

b) The connection for PNP Hall elements taken as both deceleration signal and zero signal is shown in Fig. 2-9:

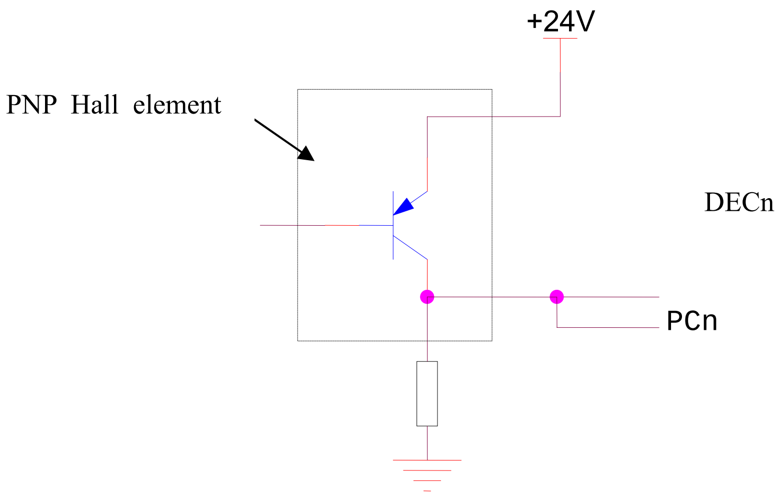


Fig 2-9 Connection using PNP Hall elements

2.1.7 Connection to drive unit

The connection of GSK 980MDa to GSK drive unit is shown in Fig. 2-10:

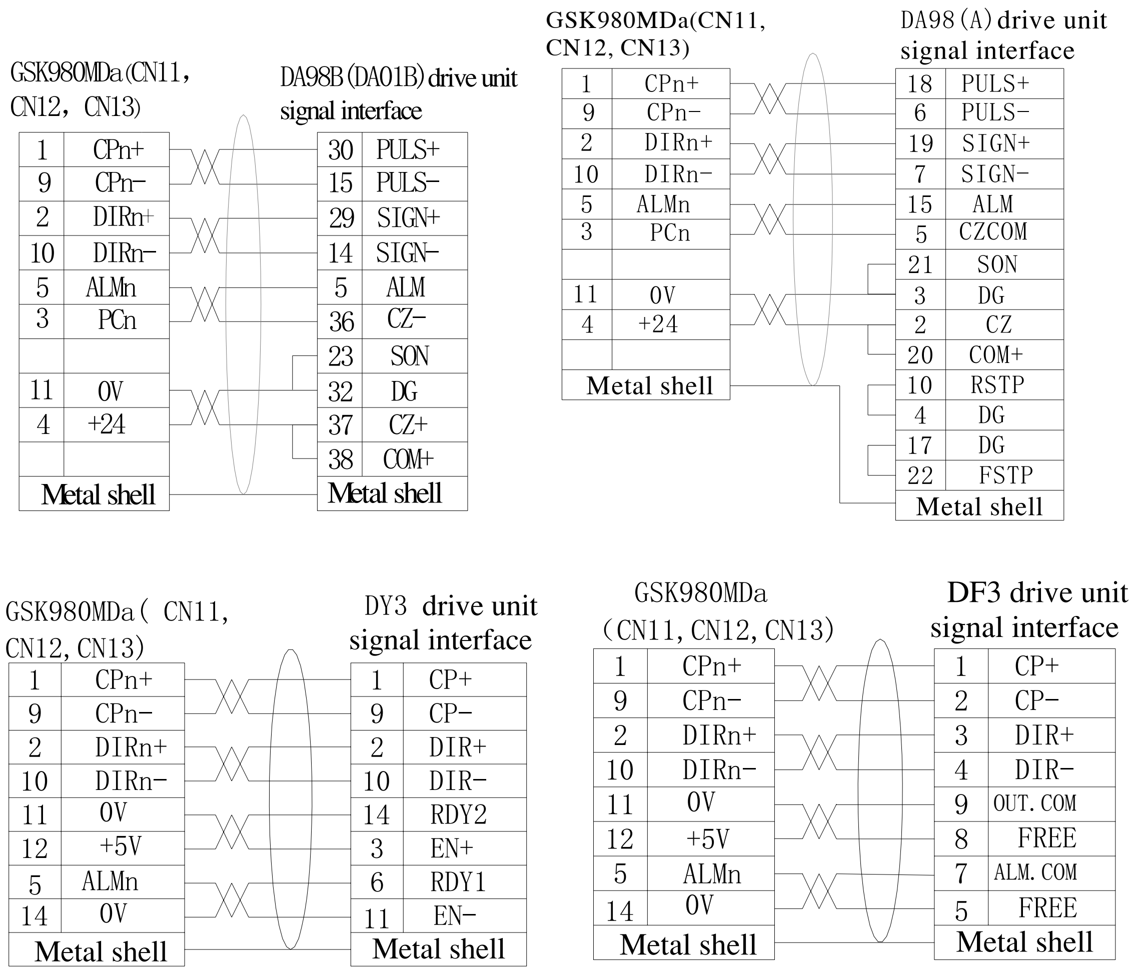
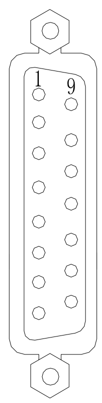


Fig.2-10 Connection of 4th axis interface to drive unit

2.2 Connection of 4th axis

2.2.1 4th axis interface definition

1：CP4+
2：DIR4+
3：PC4
4：+24V
5：ALM4
6：SET4
7：EN4
8：RDY4/ZSD4



9： CP4-
10：DIR4-
11：GND
12：VCC
13：VCC
14：GND
15：GND

Signal	Explanation
CP4+, CP4-	Command pulse signal
DIR4+, DIR4-	Command direction signal
PC4	Zero signal
ALM4	Drive alarm signal
EN4	Axis enable signal
SET4	Pulse disable signal

Fig.2-11 Interface CN14（DB15 female）

2.2.2 Connection of 4th axis interface as linear axis

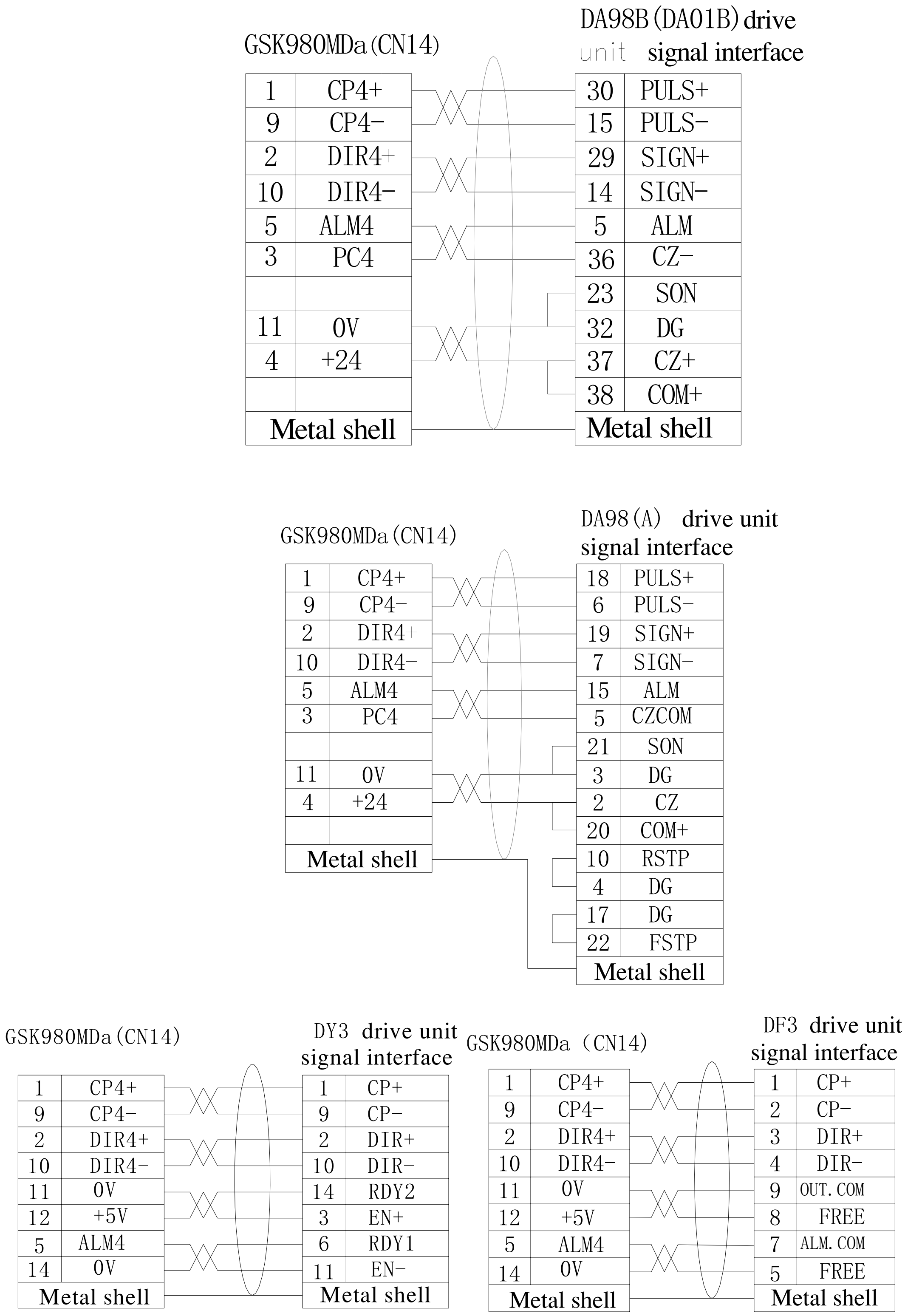


Fig.2-12 Connection of 4th axis interface to drive unit

2.2.3 Connection of 4th axis interface as rotary axis

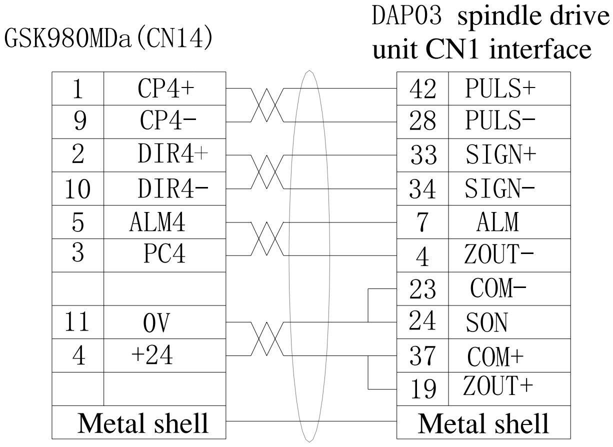


Fig.2-13 Connection of 4th axis interface to spindle drive unit

2.3 Connection of spindle port

2.3.1 Definition of signal

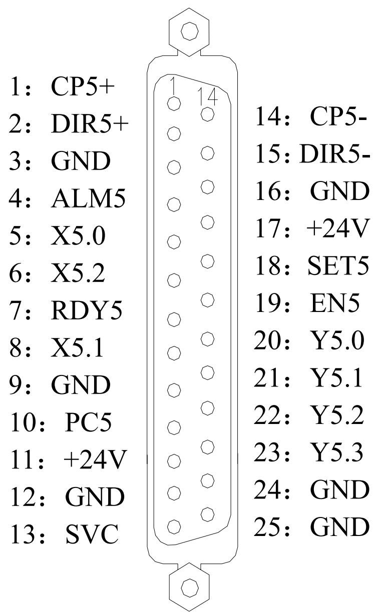


Fig.2-14 CN15 Spindle Prot

CP5+, CP5-	Spindle pulse signal
DIR5+, DIR5-	Spindle direction signal
ALM5	Spindle alarm signal
RDY5	Spindle is ready
PC5	Spindle zero signal
SVC	Output of voltage
SET5	Spindle disable signal
EN5	Spindle enable signal
X5.0~X5.2	PLC Address,only For these,Lower voltage is valid
Y5.0~Y5.3	PLC address

2.3.2 Spindle zero signal

Except for the PC5 signal, other fixed signals of the spindle interface are the same as that of the X,Y,Z, 4th axes. the PC5 interface circuit is shown as follows:

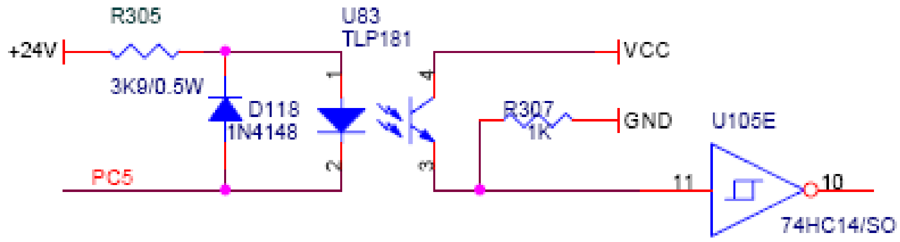


Fig.2-15 Spindle zero signal interface circuit

2.3.3 Linear axis

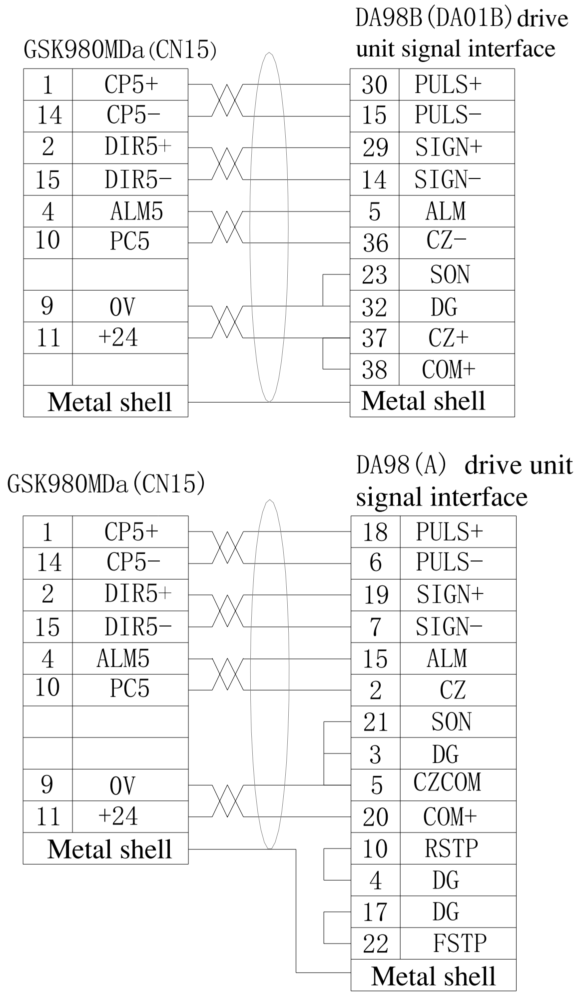


Fig.2-16 Connection of spindle interface to drive unit

2.3.4 Connected with inverter

The connection of GSK980MDa with convertor is shown in Fig. 2-17:

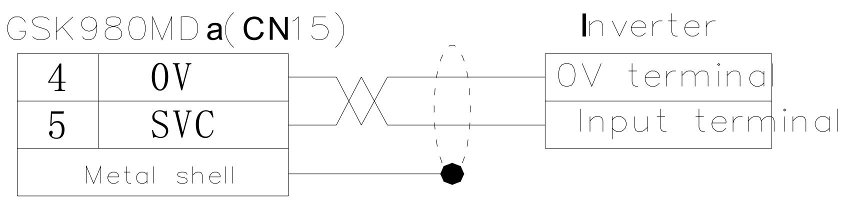


Fig.2-17 Connection of GSK980MDa to inverter

2.3.5 Connection of spindle interface as rotary axis

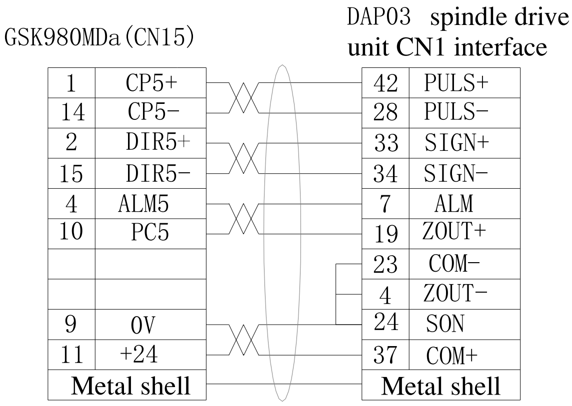


Fig.2-18 Connection of spindle to DAP03

2.3.6 Connection of spindle interface as "CS" axis

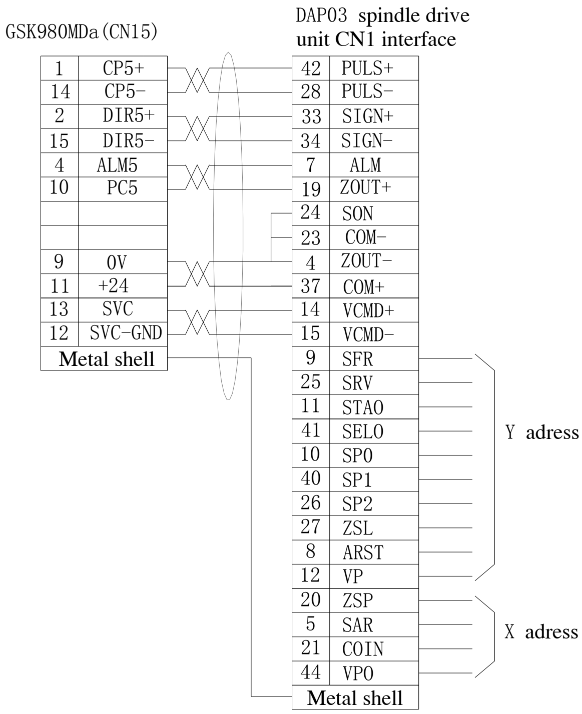


Fig.2-19 Connection of spindle to DAP03

2.3.7 SVC Signal explanation

The analog spindle interface SVC can output 0~10V voltage, its interior signal circuit is shown in Fig. 2-20:

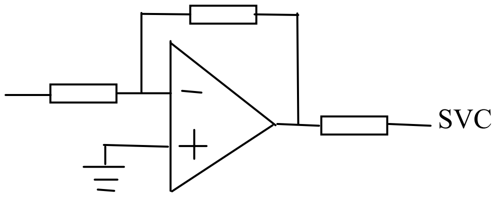
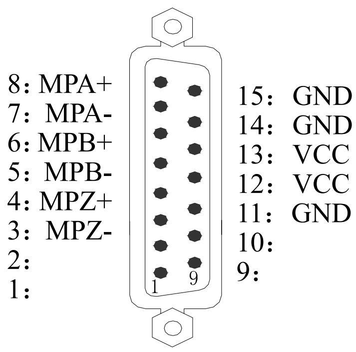


Fig 2-20 SVC Signal circuit

2.4 Connection to Spindle Encoder

2.4.1 Spindle encoder interface definition



Name	Explanation
MPA-/MPA+	Encode A phase pulse
MPB-/MPB+	Encode B phase pulse
MPZ-/MPZ+	Encode Z phase pulse

Fig.2-21 CN21 Encode interface
（DB15 male socket）

2.4.2 Signal Explanation

MPZ-/MPZ+, MPB-/MPB+, MPA-/MPA+ are the encoder Z, B, A phase differential input signals respectively, which are received by 26LS32; MPB-/MPB+, MPA-/MPA+ are normal square wave of phase shift 90°with the maximum signal frequency less than 1MHz; the encoder pulses for GSK980MDa are set by data parameter No.109, whose range is from 100 to 5000.

Its interior connection circuit is shown in Fig. 2-22: (n=A, B, C)

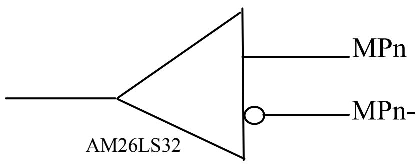


Fig.2-22 Encode signal circuit

2.4.3 Connection of spindle encoder interface

The connection of GSK980MDa to spindle encoder is shown in Fig. 2-23, twisted pair cables are used to connection.

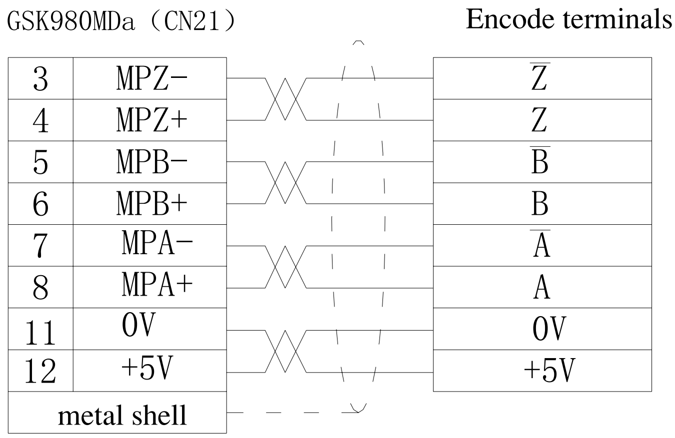


Fig.2-23 Connection of GSK980MDa to encoder

2.5 Connection to Handwheel

2.5.1 Handwheel interface definition

13：GND
12：GND
11：GND
10：GND
9：X6.3
8：X6.2
7：
6：X6.1
5：X6.0
4：HB-
3：HB+
2：HA-
1：HA+

26：
25：
24：
23：X6.5
22：X6.4
21：
20：
19：
18：+24V
17：+24V
16：+5V
15：+5V
14：+5V

Signal	Explanation
HA+, HA-	Handwheel A phase signal
HB+, HB-	Handwheel B phase signal
X6.0~X6.5	PLC adress
+24V	Direct current
VCC, GND	

Fig.2-24 CN31 handwheel interface
（3-line DB26 male socket）

2.5.2 Signal explanation

“HA+”, ”HA-“, ”HB+”, ”HB-“ are the input singals of handwheel A and B phases. Its interior connection circuit is shown in Fig. 2-25:

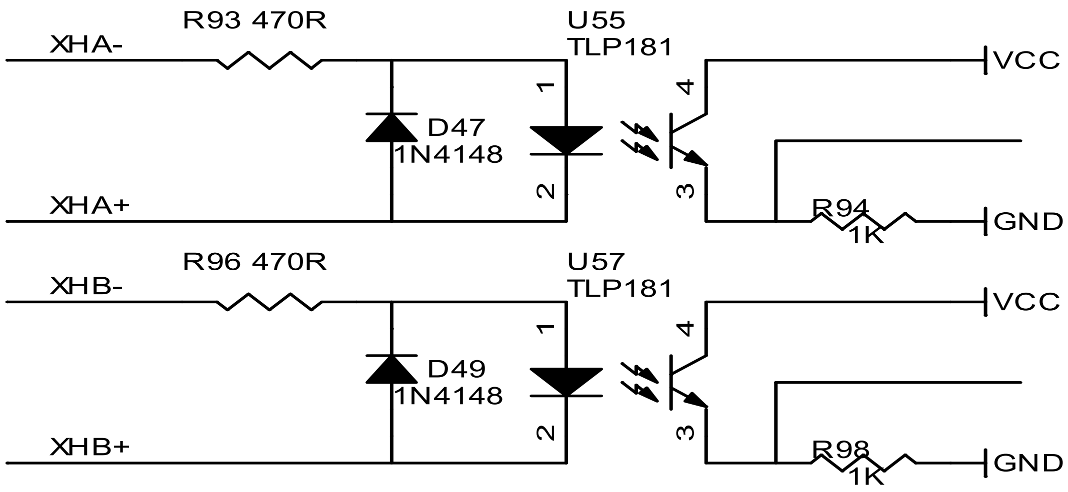


Fig.2-25 Handwheel signal circuit

The connection of GSK980MDa to handwheel is shown in Fig. 2-26：

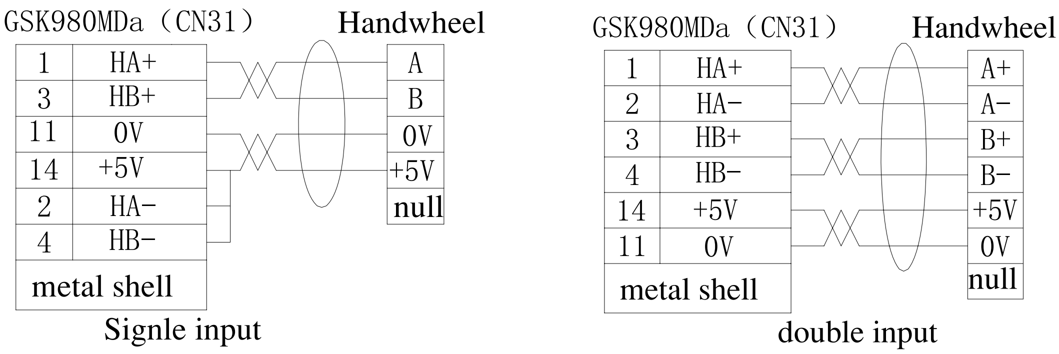
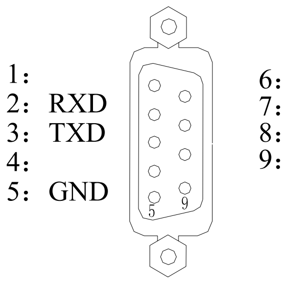


Fig.2-26 Connection of GSK980MDa to handwheel

2.6 Connection of GSK980MDa to PC

2.6.1 Communication interface definition



Signal	Explanation
RXD	For date reception
TXD	For date transmiting
GND	For signal grounding

Fig.2-27　CN51 communication interface
(DB9 female socket)

2.6.2 Communication interface connection

The communication between GSK980MDa and PC can be done via RS232 interface (GSK980MDa communication software needed), The connection of them is shown in Fig.2-28

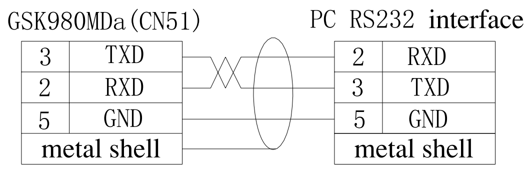


Fig.2-28 Connection of GSK980MDa to PC

The communication of a GSK980MDa to another GSK980MDa can be made via their CN51 interfaces, and the connection of them is shown in Fig.2-29:

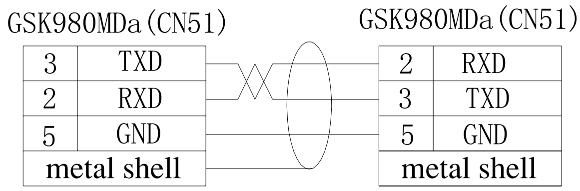


Fig.2-29 Communication connection of GSK980MDa to GSK980MDa

2.7 Connection of Power Interface

GSK-PB2 power box is applied in this GSK980MDa, which involves 4 groups of voltage: +5V （3A）, +12V（1A）, -12V（0.5A）, +24V（0.5A）, and its commom terminal is COM（0V）. The connection of GSK-PB2 power box to GSK980MDa CN1 interface has been done for its delivery from factory, and the user only need to connect it to a 220V AC power in using:

The interface definition of GSK980MDa CN1 is shown below:

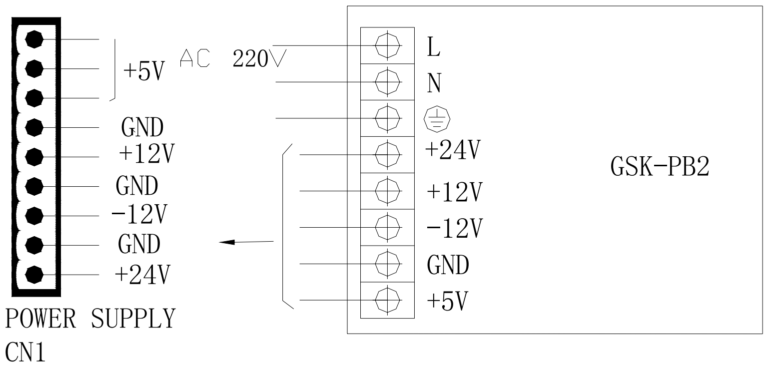


Fig.2-30

2.8 I/O Interface Definition：

CN61：44-core (3-line) male socket

NO.	Address	NO.	Address	NO.	Address	NO.	Address
1	X0.0	12	X1.3（DECZ）	23	GND	34	X2.5(DEC5)
2	X0.1	13	X1.4	24	GND	35	X2.6
3	X0.2	14	X1.5	25		36	X2.7
4	X0.3（DECX）	15	X1.6	26		37	X3.0
5	X0.4	16	X1.7	27		38	X3.1
6	X0.5（ESP）	17		28		39	X3.2
7	X0.6	18		29	X2.0	40	X3.3
8	X0.7	19		30	X2.1	41	X3.4
9	X1.0	20		31	X2.2	42	X3.5（SKIP）
10	X1.1	21	GND	32	X2.3（DECY）	43	X3.6
11	X1.2	22	GND	33	X2.4（DEC4）	44	X3.7

CN62：44-core (3-line) female socket

NO.	Address	NO.	Address	NO.	Address	NO.	Address
1	Y0.0	12	Y1.3	23	+24V	34	Y2.5
2	Y0.1	13	Y1.4	24	+24V	35	Y2.6
3	Y0.2	14	Y1.5	25	+24V	36	Y2.7
4	Y0.3	15	Y1.6	26	GND	37	Y3.0
5	Y0.4	16	Y1.7	27	GND	38	Y3.1
6	Y0.5	17	GND	28	GND	39	Y3.2
7	Y0.6	18	GND	29	Y2.0	40	Y3.3
8	Y0.7	19	GND	30	Y2.1	41	Y3.4
9	Y1.0	20	+24V	31	Y2.2	42	Y3.5
10	Y1.1	21	+24V	32	Y2.3	43	Y3.6
11	Y1.2	22	+24V	33	Y2.4	44	Y3.7

Note 1: The I/O function of GSK980MDa drilling and milling CNC is defined by ladder diagram;

Note 2:If output function is valid, the output signal is on to 0V. If output function is invalid, the output signal is cut off by high impendance;

Note 3: If input function is valid, the input signal is on to 24V. If input function is invalid, the input signal is cut off with it;

Note 4: The effectiveness of +24V, 0V is equal to GSK980MD power box terminals that have the same name;

Note 5: XDEC, YDEC, ZDEC, DEC4, DEC5, ESP, SKIP are fixed signals that can't be altered.

2.8.1 Input Signal

Input signal means the signal from machine to CNC, when this signal is on with +24V, the input is valid; when it is off with +24V, the input is invalid. The contact point of input signal at machine side should meet the following conditions:

The capacity of the contact point: DC30V, 16mA above

Leakage current between contact points in open circuit: 1mA below

Voltage drop between contact points in closed circuit: 2V below (current 8.5mA, including cable voltage drop)

There are two external input types for input signals: one type is input by trigger point switch whose signals are from keys, stroke switch and contacts of relay at machine side, as is shown in Fig 2-31:

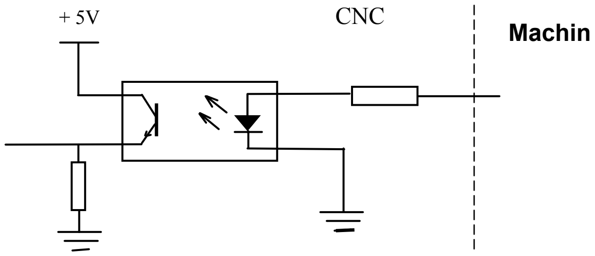


Fig.2-31

The other type is input by switch with no contacts (transistor), as is shown in Fig. 2-32, 2-33

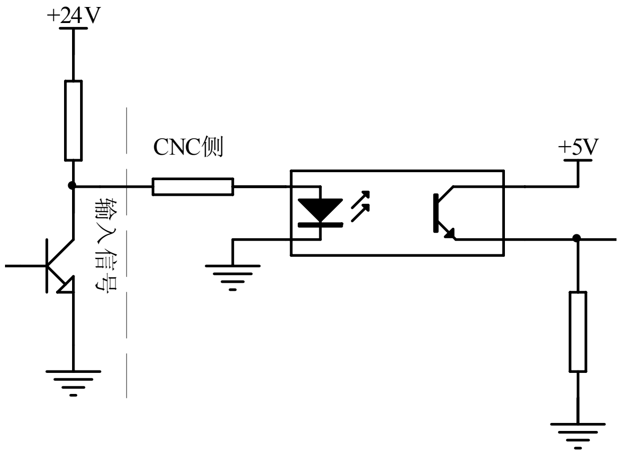


Fig.2-32 Connection of NPN

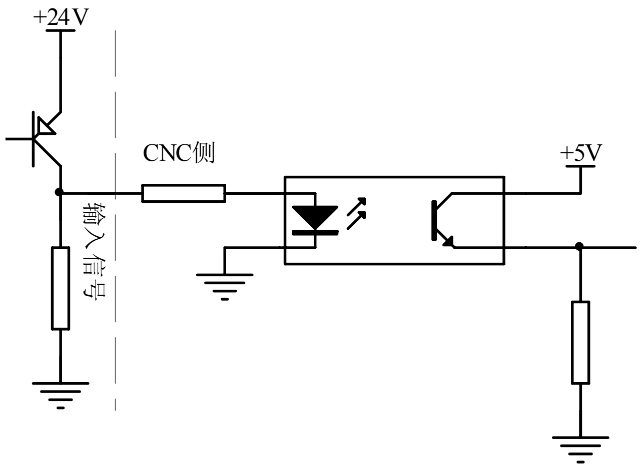


Fig.2-33 Connection of PNP

2.8.2 Output signal

The output signal is used for the machne relay and indicator, if it is on with 0V, the output function is valid; if it is off with 0V, the output function is invalid. There are total 36 digital volume outputs in I/O interface that they all have the same structure as is shown in Fig.2-34:

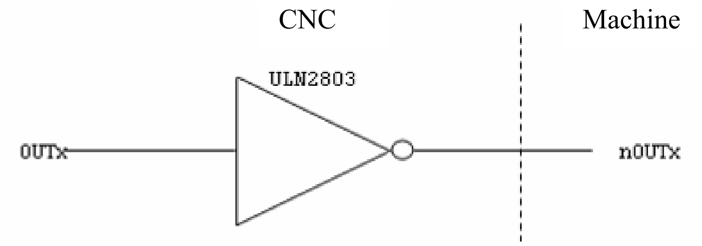


Fig.2-34 Circuit for digital volume output module

The logic signal OUTx output from the main board is sent to the input terminal of inverter (ULN2803) via a connector. And there are 2 output types for nOUTx: output with 0V, or high impedance. Its typical application is shown in follows:

- To drive LED

  A serial resistance is needed to limit the current (usually 10mA) that goes through the LED by using ULN2803 output to drive LED, which is shown in Fig.2-35

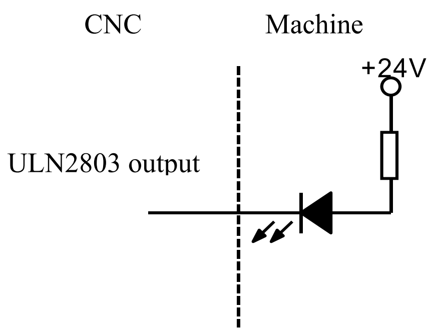


Fig.2-35

- To drive filament indicator

  An external preheat resistance is needed to decrease the current impact at power on by using ULN2803 output to drive filament indicator, and this resistance value should be within a range that the indicator cann't light up. It is shown in Fig.2-36:

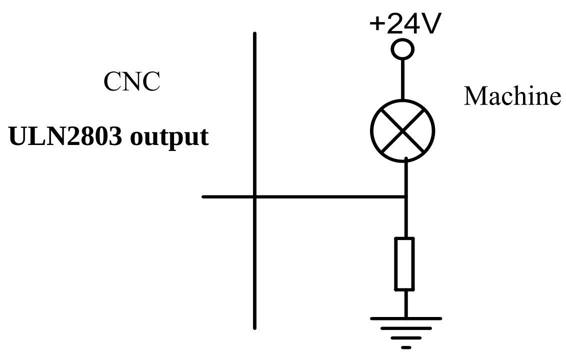


Fig. 2-36

- To drive inductive load (relay etc.)

To use ULN2803 output to drive an inductive load, it requires to connect a freewheeling diode near the coil to protect output circuit and deduce interference. It is shown in Fig.2-37:

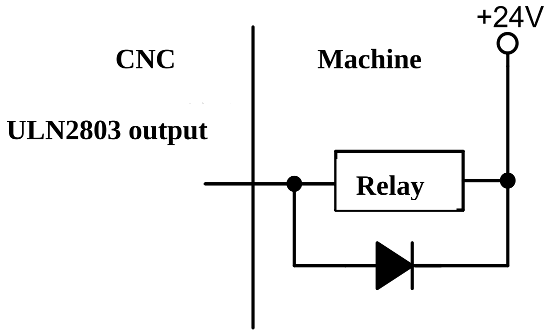


Fig.2-37

2.9 Machine Zero

- Relative signal

DECX	X axis deceleration signal	PCX	X axis zero signal
DECY	Y axis deceleration signal	PCY	Y axis zero signal
DECZ	Z axis deceleration signal	PCZ	Z axis zero signal
DEC4	4th axis deceleration signal	PC4	4th axis zero signal
DEC5	5th axis deceleration signal	PC5	5th axis zero signal

- CNC diagnosis

0 0 0								
				DEC5	DEC4	DECZ	DECY	DECX
Corresponding pin-out				CN61.34	CN61.33	CN61.12	CN61.32	CN61.4
PLC address				X2.5	X2.4	X1.3	X2.3	X0.3

0 0 8								
				PC5	PC4	PCZ	PCY	PCX
Corresponding pin-out				CN15.10	CN14.3	CN13.3	CN12.3	CN11.3

- Bit parameter

0 0 4								
			DECI					

DECI =1: Deceleration signal is on with 24V for deceleration when machine zero return is performed

=0: Deceleration signal is off 24V for deceleration when machine zero return is performed

0 0 6								
				ZM5	ZM4	ZMZ	ZMY	ZMX

ZMX =1：X axis machine zero return type C;

=0：X axis machine zero return type B.

ZMY =1：Y axis machine zero return type C;

=0：Y axis machine zero return type B.

ZMZ =1：Z axis machine zero return type C;

=0：Z axis machine zero return type B.

ZM4 =1：4th axis machine zero return type C;

=0：4th axis machine zero return type B.

ZM5 =1：5th axis machine zero return type C;

=0：5th axis machine zero return type B.

0	0	7

			ZC5	ZC4	ZCZ	ZCY	ZCX

ZCX =1：The deceleration signal （DECX）and one-rotation signal （PCX）of X axis are in parallel connection during machine zero return (a proximity switch acting as both the deceleration signal and zero signal);

=0：The deceleration signal（DECX）and one-rotation signal（PCX）of X axis are connected independently during machine zero return (the indepent deceleration signal and zero signal are required) .

ZCY =1：The deceleration signal （DECY）and one-rotation signal （PCY）of Y axis are in parallel connection during machine zero return (a proximity switch acting as both the deceleration signal and zero signal);

=0：The deceleration signal（DECY）and one-rotation signal（PCY）of Y axis are connected independently during machine zero return (the indepent deceleration signal and zero signal are required) .

ZCZ =1：The deceleration signal （DECZ） and one-rotation signal （PCZ）of Z axis are in parallel connection during machine zero return (a proximity switch acting as both the deceleration signal and zero signal);

=0：The deceleration signal（DECZ） and one-rotation signal（PCZ）of Z axis are connected independently during machine zero return (the indepent deceleration signal and zero signal are required) .

ZC4 =1：The deceleration signal（DEC4） and one-rotation signal（PC4）of 4th axis are in parallel connection during machine zero return (a proximity switch acting as both the deceleration signal and zero signal);

=0：The deceleration signal（DEC4） and one-rotation signal（PC4）of 4th axis are connected independently during machine zero return (the indepent deceleration signal and zero signal are required) .

ZC5 =1：The deceleration signal（DEC5） and one-rotation signal（PC5）of 5th axis are in parallel connection during machine zero return (an proximity switch acting as both the deceleration signal and zero signal);

=0：The deceleration signal（DEC5） and one-rotation signal（PCZ）of 5th axis are connected

independently during machine zero return (the indepent deceleration signal and zero signal are required) .

0	1	1						ZNIK		

ZNLK =1：The direction keys are locked as machine zero return is performed,by pressing the direction key once,it moves to the machine zero automatically and stops,By pressing the key at the machine zero return,the motion stops immediately;

=0：The direction keys are not locked as machine zero return is performed, but the direction keys should be pressed and held on

0	1	2								ISOT

ISOT =1：Manual rapid traverse valid prior to machine zero return;

=0：Manual rapid traverse invalid prior to machine zero return.

0	1	4				ZRS5	ZRS4	ZRSZ	ZRSY	ZRSX

ZRSZ, ZRSX, ZRSY, ZRS4, ZRS5 =1: To select machine zero return type B, C, which have machine zero, it needs to detect deceleration and zero signals in machine zero return;

=0: To select machine zero return type A, which has no machine zero, it does not detect deceleration and zero signals in machine zero return.

0	2	2				MZR5	MZR4	MZRZ	MZRY	MZRX

MZRX, MZRZ, MZRY, MZR4, MZR5 =1：The direction of zero return is negative for X, Z, Y ,4th,5th axes;

=0：The direction of zero return is positive for X, Z, Y,4th ,5th axes

- Date parameter

089	Low speed of machine zero return of X axis
090	Low speed of machine zero return of Y axis
091	Low speed of machine zero return of Z axis
092	Low speed of machine zero return of 4th axis
093	Low speed of machine zero return of 5th axis

094	High speed of machine zero return of X axis
095	High speed of machine zero return of Y axis
096	High speed of machine zero return of Z axis
097	High speed of machine zero return of 4th axis
098	High speed of machine zero return of 5th axis

130	X axis machine zero offset (0.001)
131	Y axis machine zero offset (0.001)
132	Z axis machine zero offset (0.001)
133	The 4th axis machine zero offset (0.001)
134	The 5th axis machine zero offset (0.001)

145	X machine coordinate of the 1^{st} reference point (0.001mm)
146	Y machine coordinate of the 1^{st} reference point (0.001mm)
147	Z machine coordinate of 1^{st} reference point (0.001mm)
148	4^{th} machine coordinate of the 1^{st} reference point (0.001mm)
149	5^{th} machine coordinate of the 1^{st} reference point (0.001mm)

150	X machine coordinate of the 2^{nd} reference point (0.001mm)
151	Y machine coordinate of the 2^{nd} reference point (0.001mm)
152	Z machine coordinate of the 2^{nd} reference point (0.001mm)
153	4^{th} machine coordinate of the 2^{nd} reference point (0.001mm)
154	5^{th} machine coordinate of the 2^{nd} reference point (0.001mm)

155	X machine coordinate of the 3rd reference point (0.001mm)
156	Y machine coordinate of the 3rd reference point (0.001mm)
157	Z machine coordinate of the 3rd reference point (0.001mm)
158	4^{th} machine coordinate of the 3rd reference point (0.001mm)
159	5^{th} machine coordinate of the 3rd reference point (0.001mm)

160	X machine coordinate of the 4th reference point (0.001mm)
161	Y machine coordinate of the 4th reference point (0.001mm)
162	Z machine coordinate of the 4th reference point (0.001mm)
163	4^{th} machine coordinate of the 4th reference point (0.001mm)
164	5^{th} machine coordinate of the 4th reference point (0.001mm)

- **Signal connection**

  The interior wiring circuit of deceleration signal is shown in Fig.2-37

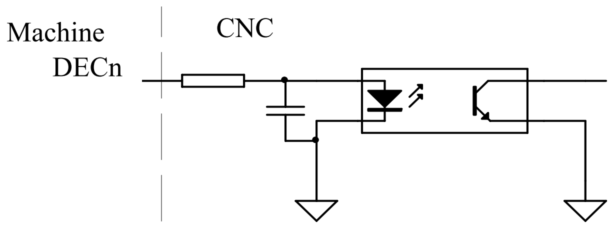
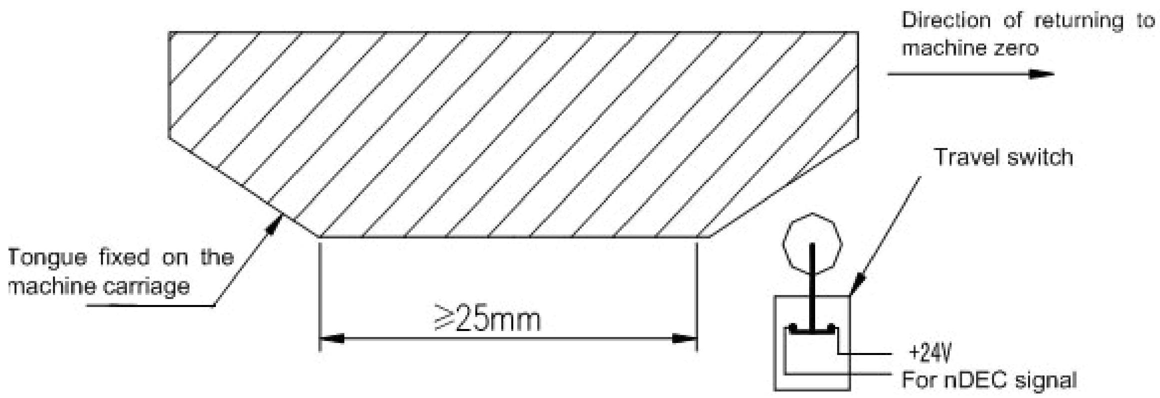


Fig.2-37

- **achine zero return type B by regarding servo motor one-rotation signal as zero signal**

  ①Its sketch map is shown in follows:



② The circuit of deceleration signal (for three axes)

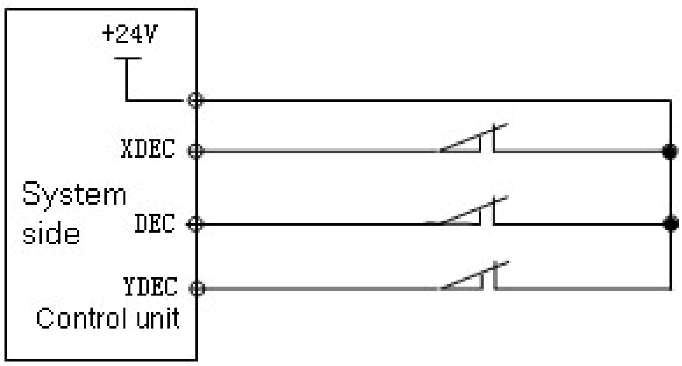


Fig.2-40

③ Action time sequence of machine zero return

When ZMn(n is X,Y,Z,4th,5th axis) of the bit parameter No.006, ZCn(n=X, Y, Z, 4th, 5th) of bit parameter No.007 and the BIT5（DECI）of the bit parameter No.004 are all set to 0, the deceleration signal low level is valid. The action time sequence of machine zero return is shown in follows

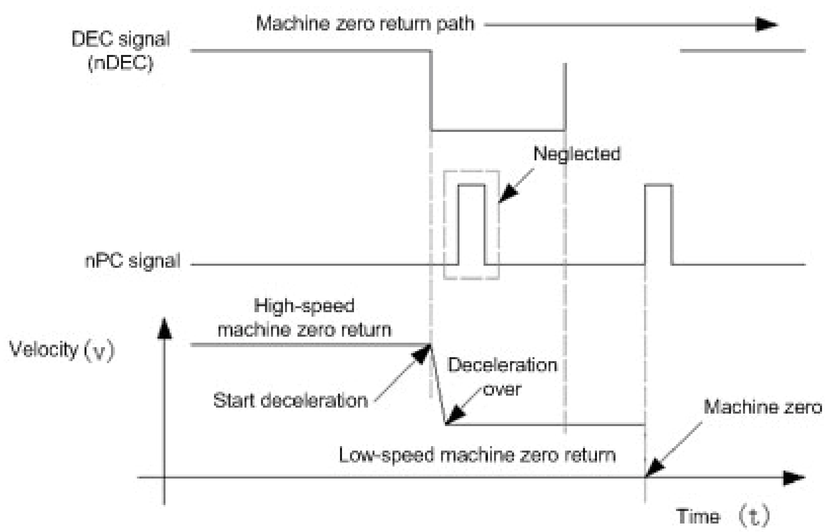


Fig.2-41

④Machine zero return process

A：Select machine zero return mode, press the manual positive or negative feed key(machine zero return direction is set by bit parameter No.022), the corresponding axis moves to the machine zero by a rapid traverse speed. As the axis press down the deceleration switch to cut off deceleration signal, the feed slows down immediately, and it continues to run in a fixed low speed.

B : When the deceleration switch is released, the deceleration signal contact point is closed again. And CNC begins to detect the encoder one-rotation signal, if the signal level changes, the motion will be stoped. And the corresponding zero indicator on the operator panel lights up for machine zero return completion

- **Machine zero return type B as an proximity switch is taken as both deceleration and zero signals**

① Its sketch map is shown in follows:

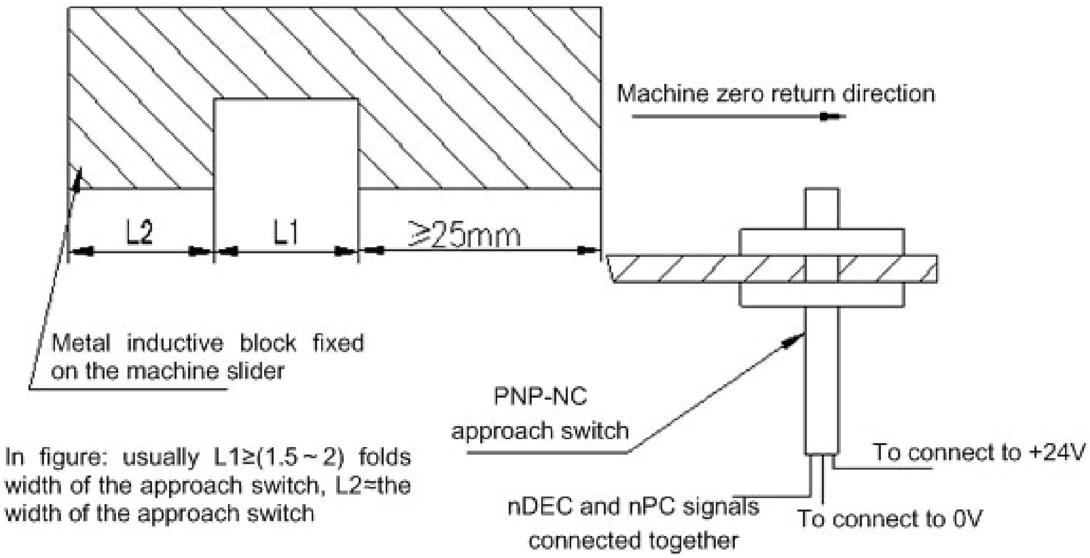


Fig.2-42

② Wiring of the deceleration signal

See details in Section 2.1.6 of this chapter

③ Action time sequence of machine zero return

When ZMn (n is X,Y,Z,4th ,5th axis)of the bit parameter No.006 and the BIT5（DECI）of the bit parameter No.004 are all set to 0, ZCn (n is X,Y,Z,4th ,5th axis)of the bit parameter No.007 is set to 1, the deceleration signal low level is valid . The action time sequence of zero return is shown in follows:

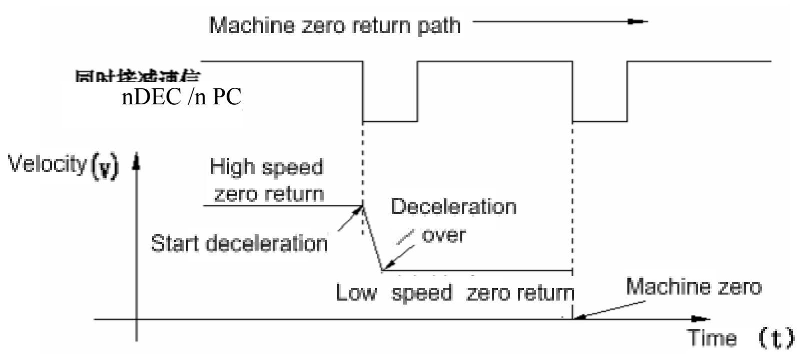


Fig.2-43 the action time sequence of zero return

④ Machine zero returns process

A：Select the Machine Zero mode, press manual positive or negative (zero return direction set by bit parameter No.183) feed key, the corresponding axis will move to the zero at a traverse speed.

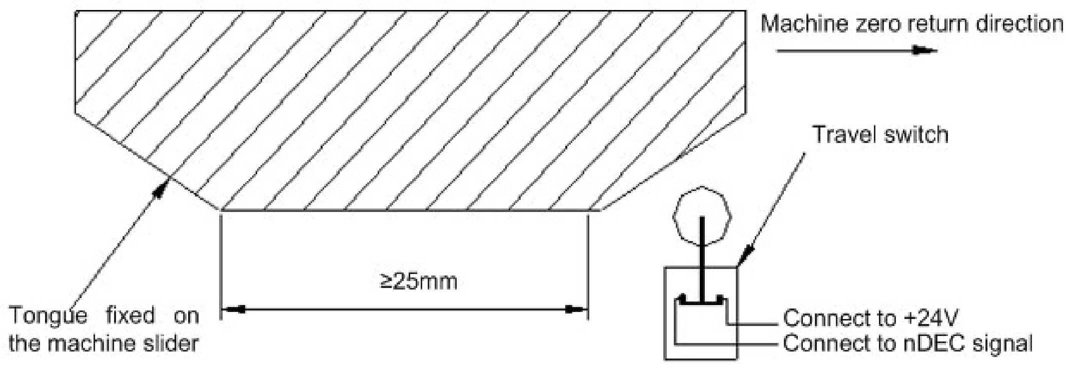
B：As the approach switch touches the tongue for the first time, the deceleration signal is valid and it slows down immediately to run in a low speed.

C：As the approach switch detaches the tongue, the deceleration signal is invalid, it moves at a fixed low speed after deceleration and starts to detect zero signal (PC).

D：As the approach switch touches the tongue for the second time, the zero signal is valid and the movement stops. The indicator for zero return on the panel lights up.

- **Machine zero return type C as servo motor one-rotation signal taken as zero signal**

① Its sketch map is shown below:



② Circuit of the deceleration signal

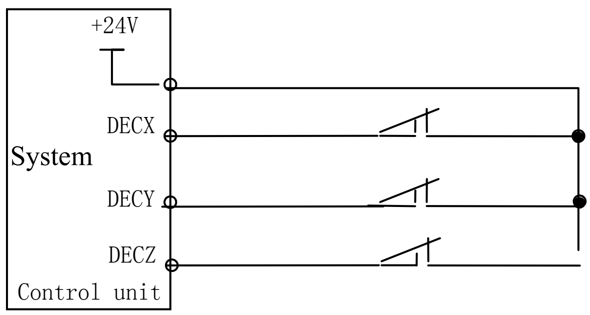


Fig.2-45

③ Action time sequence of machine zero return

When ZMn (n is X,Y,Z,4th ,5th axis) of the bit parameter No.006 are all set for 1, ZCn (n is X,Y,Z,4th ,5th axis)of the bit parameter No.007 are all set for 0, the BIT5（DECI）of the bit parameter No.004 is set for 0, and the deceleration signal low level is valid. The action time sequence of machine zero return is shown in follows

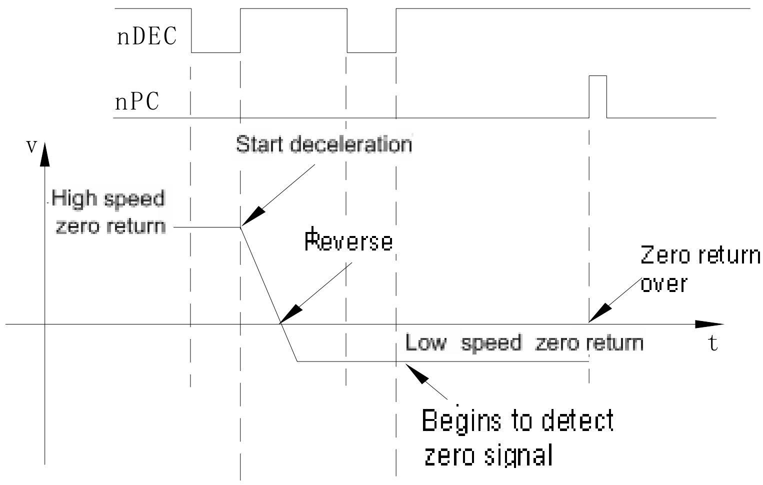


Fig.2-46

④ Machine zero returns process

A : Select the Machine Zero mode, press manual positive or negative (zero return direction set by bit parameter №022) feed key, the corresponding axis will move to the machine zero at a traverse speed. Then it touches the tongue and presses down the deceleration switch, and moves forward. When the tongue detaches the deceleration switch, the axis slows down to zero, then moves reversely and accelerates to a fixed low speed for continuous moving

B：As the tongue touches the deceleration switch for the second time, it moves on till the tongue detaches the deceleration switch. And it begins to detect the zero signals. If the zero signal level changes, the movement stops. Then zero return indicator of the corresponding axis on the panel lights up and machine zero operation is finished.

- **Machine zero return type C as an proximity switch is taken as both deceleration and zero signals**

① Its sketch map is shown below:

Volume Ⅲ Installation

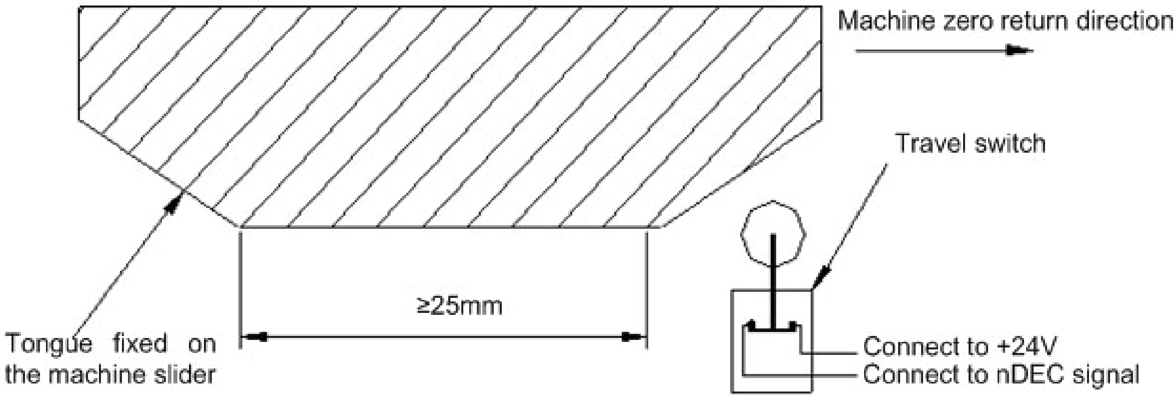


Fig.2-47

② Circuit of the deceleration signal

See details in Section 2.1.6 of this chapter

③ Action time sequence of machine zero return

When ZMn (n is X,Y,Z,4th ,5th axis) of the bit parameter No.006 and ZCn (n is X,Y,Z,4th ,5th axis)of the bit parameter No.007 are all set to 1, the BIT5（DECI）of the bit parameter No.004 is set to 0, the deceleration signal low level is valid. The action time sequence of machine zero return is shown in follows:

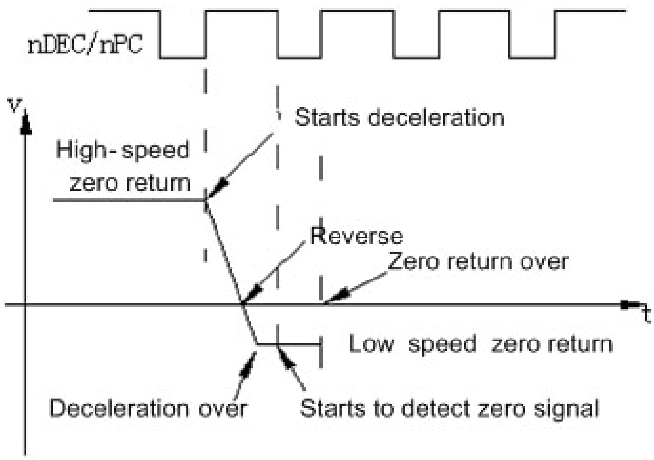


Fig.2-48

④ Machine zero returns process

A：Select the Machine Zero mode, press manual positive or negative (zero return direction is set by bit parameter No.183) feed key, the corresponding axis will move to the machine zero at a traverse speed. Then it touches the tongue and presses down the deceleration switch, and moves forward. When the tongue detaches the deceleration switch, the axis slows down to zero speed, then moves reversely and accelerates to a fixed low speed for continuous moving

B：As the tongue touches the deceleration switch for the second time, it begins to detect the zero signal. It moves on till the tongue detaches the deceleration switch, the movement stops immediately. Then zero return indicator of the corresponding axis on the panel lights up and machine zero return operation is finished.

CHAPTER 3 PARAMETER

In this chapter the CNC bit and data parameters are introduced. Various functions can be set by these parameters.

3.1 Parameter Description (by sequence)

3.1.1 Bit parameter

The expression of bit parameter is shown in follows：

Parameter NO.			BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0

0	0	1	***	***	***	ACS	HWL	***	***	***

ACS =1: Analog voltage control of spindle speed;
=0: Switching control of spindle speed.
HWL =1: MPG mode;
=0: Step mode.

0	0	2	***	***	***	LIFJ	MDITL	LIFC	NRC	TLIF

LIFJ =1: Tool life management group skip valid;
=0: Tool life management group skip invalid.
MDITL =1: Tool life management valid in MDI mode;
=0: Tool life management invalid in MDI mode.
LIFC =1: Tool life counting type 2, by times;
=0: Tool life counting type 1, by times.
NRC =1: Tool nose radius compensation valid;
=0: Tool nose radius compensation invalid.
TLIF =1: Tool life management valid;
=0: Tool life management invalid.

0	0	3	***	***	PCOMP	***	***	***	D/R	***

PCOMP =1: Screw-pitch error compensation valid;
=0: Screw-pitch error compensation invalid.
D/R =1: Tool offset D is diameter value;
=0: Tool offset D is radius value.

0	0	4	***	RDRN	DECI	***	PROD	***	***	SCW

RDRN =1：In G00 dry run mode, speed=feedrate × speed of dry run;
=0：G00 speed = rapid override × rapid tranverse speed.
DECI =1：Deceleration signal high level for machine zero return;
=0：Deceleration signal low level for machine zero return.

PROD =1：Relative coordinate displayed in POSITION page is programming position;

=0：Relative coordinate displayed in POSITION page involving tool compensation.

SCW =1：Inch output(inch system)valid after repower;

=0：Metric output(metric system)valid after repower

The functions of metric and inch system

There are two kinds of input and output units for CNC numerical control system: metric unit, millimeter (mm) and English unit (inch).

Output increement unit is set by Bit0（SCW）of bit parameter №004 in GSK980MDa system. SCW=0 indicates that minimum command increment, parameter and screw–pitch values are in metric units; SCW=1 indicates that minimum command increment, parameter and screw–pitch values are in inches units. The setting of this parameter depends on machine tool.

G code: By selecting G20/G21 code, it is able to set whether minimum input increment values are in inch or in metric. Executing G21 indicates that minimum input increment values are in metric; and executing G20 indicates that values are in inch,

0	0	5	***	***	SMAL	M30	***	***	PPD	PCMD

SMAL =1：Spindle manual gear shift for S command;

=0：Spindle auto gear shift for S command.

M30 =1：Cursor returns to beginning after M30 execution;

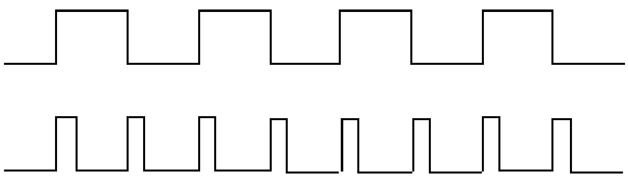
=0：Cursor not to beginning after M30 execution.

PPD =1：Relative coordinate set by G92;

=0：Relative coordinate not set by G92.

PCMD =1：Axial output wave form is pulse;

=0：Axial output wave form is square.



Square output，max. output frequency 266KPPS

Pulse output，max. output frequency 266KPPS，Pulse width 1μs.

0	0	6	***	***	***	ZM5	ZM4	ZMZ	ZMY	ZMX

ZM5 =1：5th zero return type C;

=0：5th zero return type B.

ZM4 =1：4th zero return type C;

=0：4th zero return type B.

ZMZ =1：Z zero return type C;

=0：Z zero return type B.

ZMY =1：Y zero return type C;

=0：Y zero return type B.

ZMX =1：X zero return type C;

=0：X zero return type B.

0	0	7		AVGL	***	SMZ	ZC5	ZC4	ZCZ	ZCY	ZCX

On the condition that blocks smoothing transition is valid, more smooth velocity link and better machining quality will be obtained during the path transition from line to line or from line to arc by properly changing the linear feedrate.

So the actual output speed may be different to the programming speed when using this function. And it may also differ as regard to the linear segment with the same programming speed. The deviation is not more than 15mm/min between the actual output speed and the programming speed on the condition that the programming speed F is less than 1200mm/min

AVGL =1：When SMZ=0 linear smoothing is valid,i.e. smoothing transition function is valid;
=0：Linear smoothing transition function is invalid.

SMZ =1：To execute next block till all moving blocks executed;
=0：For smooth transition between blocks.

ZC5 =1：Deceleration signal (DEC5)and one-rotation signal (PC5) of 5th axis are in parallel connection(a proximity switch taken as both deceleration signal and zero signal) during machine zero return;
=0：Deceleration signal (DEC5) and one-rotation signal (PC5) of 5th axis are connected independently (independent deceleration signal and zero signal are required) during machine zero return.

ZC4 =1：Deceleration signal (DEC4)and one-rotation signal (PC4) of 4th axis are in parallel connection (a proximity switch taken as both deceleration signal and zero signal) during machine zero return;
=0：Deceleration signal (DEC4) and one-rotation signal (PC4) of 4th axis are connected independently (independent deceleration signal and zero signal are required) during machine zero return.

ZCZ =1：Deceleration signal (DECZ) and one-rotation signal (PCZ) of Z axis are in parallel connection a proximity switch taken as both deceleration signal and zero signal) during machine zero return;
=0：Deceleration signal (DECZ) and one-rotation signal (PCZ) of Z axis are connected independently (independent deceleration signal and zero signal are required) during machine zero return.

ZCY =1：Deceleration signal (DECY) and one-rotation signal (PCY) of Y axis are in parallel connection a proximity switch taken as both deceleration signal and zero signal) during machine zero return;
=0：Deceleration signal (DECY) and one-rotation signal (PCY) of Y axis are connected independently (independent deceleration signal and zero signal are required) during machine zero return.

ZCX =1：Deceleration signal (DECX)and one-rotation signal (PCX) of X axis are in parallel connection a proximity switch taken as both deceleration signal and zero signal) during

machine zero return;

=0：Deceleration signal (DECX) and one-rotation signal (PCX) of X axis are connected independently (independent deceleration signal and zero signal are required) during machine zero return.

0	0	8		DISP	***	***	DIR5	DIR4	DIRZ	DIRY	DIRX

DISP =1：Enter absolute page after power on;

=0：Enter relative page after power on.

DIR5 =1：Direction signal (DIR)is high level as 5th axis moves positively;

=0：Direction signal (DIR)is low level as 5th axis moves negatively.

DIR4 =1：Direction signal (DIR)is high level as 4th axis moves positively;

=0：Direction signal (DIR)is low level as 4th axis moves negatively.

DIRZ =1：Direction signal (DIR)is high level as Z axis moves positively;

=0：Direction signal (DIR)is low level as Z axis moves negatively.

DIRY =1：Direction signal (DIR)is high level as Y axis moves positively;

=0：Direction signal (DIR)is low level as Y axis moves negatively.

DIRX =1：Direction signal (DIR)is high level as X axis moves positively;

=0：Direction signal (DIR)is low level as X axis moves negatively.

0	0	9		***	***	***	ALM5	ALM4	ALMZ	ALMY	ALMX

ALM5 =1：5th axis low level alarm signal (ALM5);

=0：5th axis high level alarm signal (ALM5).

ALM4 =1：4th axis low level alarm signal (ALM4);

=0：4th axis high level alarm signal (ALM4).

ALMZ =1：Z axis low level alarm signal (ALMZ);

=0：Z axis high level alarm signal (ALMZ).

ALMY =1：Y axis low level alarm signal (ALMY);

=0：Y axis high level alarm signal (ALMY).

ALMX =1：X axis low level alarm signal (ALMX);

=0：X axis high level alarm signal (ALMX).

0	1	0		CPF7	CPF6	CPF5	CPF4	CPF3	CPF2	CPF1	CPF0

CPF0～CPF7： Setting values of backlash compensation pulse frequency.

Set frequency = （2^7×CPF7+2^6×CPF6+2^5×CPF5+2^4×CPF4+2^3×CPF3+2^2×CPF2+2^1×CPF1+CPF0） Kpps

0	1	1		BDEC	BD8	***	***	***	ZNIK	***	***

BDEC =1：Backlash compensation type B, the compensation data are output by ascending type and the set frequency is invalid.;

=0：Backlash compensation type A, the compensation data are output by the set frequency (by bit parameter No.010) or 1/8 of it.

BD8 =1：Backlash compensation is done by the 1/8 of the set frequency;

=0：Backlash compensation is done by the set frequency.

ZNIK =1：Direction keys locked during zero return, homing continues to end by pressing direction key once;

=0：Direction keys unlocked but should be held on during zero return.

0	1	2

***	***	***	TMANL	***	***	EBCL	ISOT

TMANL =1：Manual tool change for T code;

=0：Auto tool change for T code.

EBCL =1：Program end sign EOB displays “;”(semicolon);

=0：Program end sign EOB displays “*”(asterisk).

ISOT =1：Prior to machine zero return after power on, manual rapid traverse valid;

=0：Prior to machine zero return after power on, manual rapid traverse invalid.

0	1	3

SCRD	G01	RSCD	***	***	***	SKPI	G31P

SCRD =1：Coordinate system holding on at power down;

=0：Coordinate system not holding on at power down, G54 coordinate system is set after power on.

G01 =1：G01 status when power on;

=0：G00 status when power on.

RSCD =1：G54 coordinate system when reset 4;

=0：Coordinate system not changed when reset.

SKPI =1：High level valid for skip signal;

=0：Low level valid for skip signal.

G31P =1：G31 immediately stops when skip signal is valid;

=0：G31 slows down to stop when skip signal is valid.

0	1	4

***	***	***	ZRS5	ZRS4	ZRSZ	ZRSY	ZRSX

ZRS5 =1: There are machine zero point in 5th axis, it detects deceleration signal and zero signal when performing machine zero return;

=0: There are no machine zero point in 5th axis, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return.

ZRS4 =1: There are machine zero point in 4th axis, it detects deceleration signal and zero signal when performing machine zero return;

=0: There are no machine zero point in 4th axis, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return.

ZRSZ =1: There are machine zero point in Z axis, it detects deceleration signal and zero signal when performing machine zero return;

=0: There are no machine zero point in Z axis, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return.

ZRSY =1: There are machine zero point in Y axis, it detects deceleration signal and zero signal when performing machine zero return;

Volume Ⅲ Installation

=0: There are no machine zero point in Y axis, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return.

ZRSX =1: There are machine zero point in X axis, it detects deceleration signal and zero signal when performing machine zero return;

=0: There are no machine zero point in X axis, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return.

0	1	5		LPTK	RPTK	NAT	BRCH	***	***	***	***

LPTK =1：Hole locating is done by cutting feed on line continuous drilling;

=0：Hole locating is done by rapid feed on line continuous drilling;

RPTH =1: Hole locating is cutting path in circle and rectangle continuous drilling;

=0：Hole locating is rapid path in circle and rectangle continuous drilling;

NAT =1 Define the range of user macro program asin, atan;

=0：Not define the range of user macro program asin, atan;

BRCH =1：Plane returning is selected by G98 and G99 in continous drilling;

=0：Plane returning is selected by G99 in continous drilling

0	1	7		***	MST	MSP	MOT	MESP	***	***	***

MST =1：External cycle start signal (ST) invalid，

=0：External cycle start signal (ST) valid.

MSP =1：External stop signal (SP) invalid，

=0：External stop signal (SP) valid with external stop switch connected, otherwise CNC shows "stop" .

MOT =1：Not detect software stroke limit;

=0：Detect software stroke limit.

MESP =1：Emergency stop invalid;

=0：Emergency stop valid.

0	1	8		***	***	***	ESCD	***	***	***	***

ESCD =1：S code off at emergency stop;

=0：S code not off at emergency stop.

0	1	9		KEY1	***	***	HNG5	HNG4	HNGZ	HNGY	HNGX

KEY1 =1：Prog. switch ON after power on;

=0：Prog. switch OFF after power on.

HNG5 =1：5th MPG:ccw:+,cw:-;

=0：5th MPG:ccw:-,cw:+.

HNG4 =1：4th MPG:ccw:+,cw:-;

=0：4th MPG:ccw:-,cw:+.

HNGZ =1：Z MPG:ccw:+,cw:-;

=0：Z MPG:ccw:-,cw:+.

HNGY =1：Y MPG:ccw:+,cw:-;

=0：Y MPG:ccw:-,cw:+.

HNGX =1：X MPG:ccw:+,cw:-;

=0：X MPG:ccw:-,cw:+.

0	2	0		SPFD	SAR	THDA	VAL5	VAL4	VALZ	VALY	VALX

SPFD =1：Cutting feed stops if spindle stops;

=0：Cutting feed not stop after spindle stop.

SAR =1：Detect spindle SAR signal prior to cutting;

=0：Not detect spindle SAR signal prior to cutting.

THDA =1：Thread machining adopts exponential acceleration and deceleration;

=0：Thread machining adopts linear acceleration and deceleration.

VAL5 =1：For 5th axis move key,↑ is positive，↓is negative;

=0：For 5th axis move key, ↓is positive，↑is negative.

VAL4 =1：For 4th axis move key,↑ is positive，↓is negative;

=0：For 4th axis move key, ↓is positive，↑is negative.

VALZ =1：For Z axis move key,↑ is positive，↓is negative;

=0：For Z axis move key, ↓is positive，↑is negative.

VALY =1：For Y axis move key,↑ is positive，↓is negative;

=0：For Y axis move key, ↓is positive，↑is negative.

VALX =1：For X axis move key, →is positive，←is negative;

=0：For X axis move key, ←is positive，→is negative

0	2	2		CALH	SOT	***	MZR5	MZR4	MZRZ	MZRY	MZRX

CALH =1：Length offset not cancelled in reference point return;

=0：Length offset cancelled in reference point return.

SOT =1：Software limit is valid after zero return at power on;

=0：Software limit is valid once power on.

MZR5 =1：Machine zero return in negative 5th axis;

=0：Machine zero return in positive 5th axis.

MZR4 =1：Machine zero return in negative 4th axis;

=0：Machine zero return in positive 4th axis.

MZRZ =1：Machine zero return in negative Z axis;

=0：Machine zero return in positive Z axis.

MZRY =1：Machine zero return in negative Y axis;

=0：Machine zero return in positive Y axis.

MZRX =1：Machine zero return in positive X axis;

=0：Machine zero return in negative X axis.

0	2	5		RTORI	***	RTPCP	***	***	RTCRG	***	***

RTORI=1：Spindle performs zero return when M29 is executed;

=0：Spindle does not perform zero return when M29 is executed.

RTPCP=1：Rigid tapping is the high-speed deep hole cycle(G73 mode);

=0：Rigid tapping is the high-speed deep hole cycle (G83 mode).

RTCRG=1：Do not wait for G61.0 to be 1 as excuting next program block after rigid tapping cancelled;

=0：Do wait for G61.0 to be 1 as excuting next program block after rigid tapping cancelled.

0	2	6

A4IS1	A4IS0	***	RCS4	***	***	ROS4	ROT4

RCS4 =1：4th Cs function is valid(power on);

=0：4th Cs function is invalid(power on).

Note: Only when the rotary axis function is valid (ROT4=1), can the RCS4 be set valid.

ROS4, ROT4：Set the type of 4th;

	Linear	**Rotary A**	**Rotary B**	**invalid**
ROT4	0	1	1	0
ROS4	0	0	1	1

A4IS1, A4IS0:Selecte increment system of 4th.

A4IS1	**A4IS0**	**Increment System of 4TH**
0	0	Same to the X, Y, Z
0	1	IS-A
1	0	IS-B
1	1	IS-C

0	2	7

***	RRT4	***	***	***	RRL4	RAB4	ROA4

RRT4 =1：Zero mode D is used on 4th rotary axis (power on);

=0：Zero mode A,B,C are used on 4th rotary axis (power on).

RRL4 =1：4th rel.coor.cycle func.is valid (power on);

=0：4th rel.coor.cycle func.is invalid(power on).

RAB4 =1：4th rotates according to symbol direction;

=0：4th rotates according to nearby rotation.

ROA4 =1：4th abs.coor.cycle func.is valid (power on);

=0：4th abs.coor.cycle func.is invalid(power on).

Note 1: Parameter ROA4 is valid for only rotary axis (ROT4=1),

Note 2: Only parameter ROA4 =1, is RAB4 valid

Note 3: Only parameter ROA4 =1, is RRL4 valid

0	2	8

A5IS1	A5IS0	***	RCS5	***	***	ROS5	ROT5

RCS5 =1：5th Cs function is valid(power on);

=0：5th Cs function is invalid(power on).

Note: Only rotary axis function is valid (ROT5=1), is RCS5 valid.

ROS5, ROT5：Set the type of 5th;

	Linear	Rotary A	Rotary B	invalid
ROT5	0	1	1	0
ROS5	0	0	1	1

A5IS1, A5IS0: Selecte increment system of 5th..

A5IS1	A5IS0	Increment System of 5TH
0	0	Same to the X, Y, Z
0	1	IS-A
1	0	IS-B
1	1	IS-C

0	2	9

***	RRT5	***	***	***	RRL5	RAB5	ROA5

RRT5 =1：Zero mode D is used on 5th rotary axis (power on);

=0：Zero mode A,B,C are used on 5th rotary axis (power on).

RRL5 =1：5th rel.coor.cycle func.is valid (power on);

=0：5th rel.coor.cycle func.is invalid(power on).

RAB5 =1：5th rotates according to symbol direction;

=0：5th rotates according to nearby rotation.

ROA5 =1：5th abs.coor.cycle func.is valid (power on);

=0：5th abs.coor.cycle func.is invalid(power on).

Note1: ROA5 is valid to only rotary axis (ROT5=1);
Note2: Only when parameter ROA4 =1, is RAB4 valid;
Note3: Only when parameter ROA4 =1, is RRL4 valid;

0	3	8

ISC	***	***	***	***	***	***	***

ISC =1：Minimum increment system is IS-C(need restart);

=0：Minimum increment system is IS-B(do not need restart).

0	3	9

***	***	***	ABP5	ABP4	ABPZ	ABPY	ABPX

ABPx =1：Output axis pulse by two right-angle intersection phases(need restart);

=0：Output axis pulse by pulse and direction (do not need restart).

0	4	0

***	***	***	***	***	L2	L1	L0

L2, L1, L0：Interface language selection:

Volume Ⅲ Installation

Language	L2	L1	L0
Chinese	0	0	0
English	0	0	1
Frence	0	1	0
Spanish	0	1	1
Germen	1	0	0
Italian	1	0	1
Russian	1	1	0
Korean	1	1	1

3.1.2 Data parameter

0	4	9	CMRX：X axis multiplier coefficient
0	5	0	CMRY：Y axis multiplier coefficient
0	5	1	CMRZ：Z axis multiplier coefficient
0	5	2	CMR4：4th axis multiplier coefficient
0	5	3	CMR5：5th axis multiplier coefficient

Setting range： 1～32767

0	5	4	CMDX：X axis frequency division coefficient
0	5	5	CMDY：Y axis frequency division coefficient
0	5	6	CMDZ：Z axis frequency division coefficient
0	5	7	CMD4：4th axis frequency division coefficient
0	5	8	CMD5：5th axis frequency division coefficient

Setting range： 1～32767

setting range： 1～32767

Electronic gear ratio formula： $\frac{CMR}{CMD}=\frac{S\times 360}{\alpha\times L}\times\frac{Z_M}{Z_D}$

S：min. command output unit

Z_M：belt wheel teeth of lead screw

α: motor rotation angle for a pulse

Z_D：Wheel teeth of motor belt

L：Screw lead

0	5	9	X axis max. rapid traverse speed
0	6	0	Y axis max. rapid traverse speed
0	6	1	Z axis max. rapid traverse speed
0	6	2	4th axis max. rapid traverse speed
0	6	3	5th axis max. rapid traverse speed

Setting range：10～99999999（Unit：mm/min）

0	6	4	Acceleration&deceleration time constant of X axis rapid traverse (ms)
0	6	5	Acceleration&deceleration time constant of Y axis rapid traverse (ms)
0	6	6	Acceleration&deceleration time constant of Z axis rapid traverse (ms)
0	6	7	Acceleration&deceleration time constant of 4th axis rapid traverse (ms)
0	6	8	Acceleration&deceleration time constant of 5th axis rapid traverse (ms)

Setting range：10～4000（Unit：ms）

0	6	9	Rapid traverse speed when rapid override is F0

Setting range：6～4000（Unit：mm/min）

0	7	0	Axes top feedrate of cutting

Setting range：10～4000（Unit：mm/min）

0	7	1	Exponential acceleration start speed and deceleration end speed in cutting feed

Setting range：0～8000（Unit：mm/min）

0	7	2	Exponential acceleration&deceleration time constant of cutting

Setting range：10～4000（Unit：ms）

0	7	3	Start speed in manual feed.

Setting range：0～8000（Unit：mm/min）

0	7	4	Exponential acceleration&deceleration time constant of manual feed

Setting range：10～4000（Unit：ms）

0	7	5	Threading axes start speed

Setting range：6～8000（Unit：mm/min）

0	7	7	Initial speed of acc.&dec.speed of CS axis

Setting range：0～5000（Unit：deg/min）

0	7	8	Acc.&dec.time constant of CS axis

Setting range：10～10000（Unit：ms）

0	8	1	Initial speed of linear acceleration/deceleration in rigid tapping

Setting range：0～5000（Unit：mm/min）

0	8	2	Linear acc.&dec. time constant in rigid tapping tool infeed

Setting range：10～10000（Unit：ms）

0	8	3	Linear acc.&dec. time constant in rigid tapping tool retract

Setting range：0～4000（Unit：ms）, 082 setting value is used when it is set to 0.

0	8	4	Override value in rigid tapping tool retract(0: override is set to 100%)

Setting range：0～200, 0: override is set to 100%

0	8	5	Tool retract amount in deep hole rigid tapping(high-speed, standard)

Setting range：0～32767000（Unit：0.001mm）

0	8	9	Low speed of X axis machine zero return
0	9	0	Low speed of Y axis machine zero return
0	9	1	Low speed of Z axis machine zero return
0	9	2	Low speed of 4th axis machine zero return
0	9	3	Low speed of 5th axis machine zero return

Setting range：10～1000（Unit：mm/min）

0	9	4	High speed of X axis machine zero return
0	9	5	High speed of Y axis machine zero return
0	9	6	High speed of Z axis machine zero return
0	9	7	High speed of 4th axis machine zero return
0	9	8	High speed of 5th axis machine zero return

Setting range：10～921571875（Unit：mm/min）

0	9	9	Voltage compensation for 0V analog voltage output

Setting range：-1000～1000（Unit：mV）

1	0	0	Voltage offset value when spindle max. speed analog voltage 10V output

Setting range：-2000～2000（Unit：mV）

1	0	1	Max spindle speed of 1st gear when analog voltage output is 10V
1	0	2	Max.spindle speed of 2nd gear when analog voltage output is 10V
1	0	3	Max.spindle speed of 3rd gear when analog voltage output is 10V
1	0	4	Max.spindle speed of 4th gear when analog voltage output is 10V

Setting range：10～9999（Unit：r/min）

1	0	7	Spindle speed resches to signal detection delay time

Setting range：0～4080（Unit：ms）

1	0	8	Max. spindle speed fluctuation allowed by system

Setting range：50～1000（Unit：r/min）

1	0	9	spindle encoder pulses

Setting range：0～5000（Unit：p/r）, It is drilling holes when 0 indicates G74 and G84 cycle.

1	1	0	Transmission ratio of encoder and spindle- spindle gear teeth
1	1	1	Transmission ratio of encoder and spindle- encoder gear teeth

Setting range：1～255

1	1	5	X axis backlash offset
1	1	6	Y axis backlash offset
1	1	7	Z axis backlash offset
1	1	8	4^{th} axis backlash offset
1	1	9	5^{th} axis backlash offset

Setting range：0～2000(Unit:0.001mm)

1	2	0	Interval of X axis screw-pitch error compensation
1	2	1	Interval of Y axis screw-pitch error compensation
1	2	2	Interval of Z axis screw-pitch error compensation
1	2	3	Interval of 4^{th} axis screw-pitch error compensation
1	2	4	Interval of 5^{th} axis screw-pitch error compensation

Setting range：10000～999999 (Unit:0.001mm)

1	2	5	Screw-pitch error compensation position number of X axis machine zero
1	2	6	Screw-pitch error compensation position number of Y axis machine zero
1	2	7	Screw-pitch error compensation position number of Z axis machine zero
1	2	8	Screw-pitch error compensation position number of 4^{th} axis machine zero
1	2	9	Screw-pitch error compensation position number of 5^{th} axis machine zero

Setting range：0～255

1	3	0	X axis machine zero offset
1	3	1	Y axis machine zero offset
1	3	2	Z axis machine zero offset
1	3	3	4^{th} axis machine zero offset
1	3	4	5^{th} axis machine zero offset

Setting range：-99999～99999 (Unit:0.001mm)

135	Max. X coordinate value of software limit
136	Max. Y coordinate value of software limit
137	Max. Z coordinate value of software limit
138	Max. 4th coordinate value of software limit
139	Max. 5th coordinate value of software limit
140	Min. X coordinate value of software limit
141	Min. Y coordinate value of software limit
142	Min. Z coordinate value of software limit
143	Min. 4th coordinate value of software limit
144	Min. 5th coordinate value of software limit

Setting range：-9999999～+9999999 (Unit:0.001mm)

145	X machine coordinate of 1st reference point
146	Y machine coordinate of 1st reference point
147	Z machine coordinate of 1st reference point
148	4th machine coordinate of 1st reference point
149	5th machine coordinate of 1st reference point
150	X machine coordinate of 2nd reference point
151	Y machine coordinate of 2nd reference point
152	Z machine coordinate of 2nd reference point
153	4th machine coordinate of 2nd reference point
154	5th machine coordinate of 2nd reference point
155	X machine coordinate of 3rd reference point
156	Y machine coordinate of 3rd reference point
157	Z machine coordinate of 3rd reference point
158	4th machine coordinate of 3rd reference point
159	5th machine coordinate of 3rd reference point
160	X machine coordinate of 4th reference point
161	Y machine coordinate of 4th reference point
162	Z machine coordinate of 4th reference point
163	4th machine coordinate of 4th reference point
164	5th machine coordinate of 4th reference point

Setting range：-9999999～+9999999 (Unit:0.001mm)

172	Initial value of cutting feedrate when power on

Setting range：10～15000 (Unit:mm/min)

174	Feedrate of dry run

Setting range：10～99999999 (Unit:mm/min)

1	7	5	Arc radius error limit

Setting range：0～1000 (Unit:0.001mm), On arc code （G02,G03）, if error exceeds the difference excuting limit between initial point radius and end point radius, alarm will be issued.

1	7	6	Retraction amount of G73 high deep hole drilling cycle

Setting range：0～32767000 (Unit:0.001mm),

1	7	7	Cutting initial point of G83 high deep hole drilling cycle

Setting range：0～32767000 (Unit:0.001mm),

1	7	8	G110,G111,G134,G135 Lead of helical tool infeed

Setting range：0～999999（unit 0.001mm）

If setting value is less than 10, helical feeding is invalid for rough milling command G110, G111, G134, G135, and it feeds by linear type.

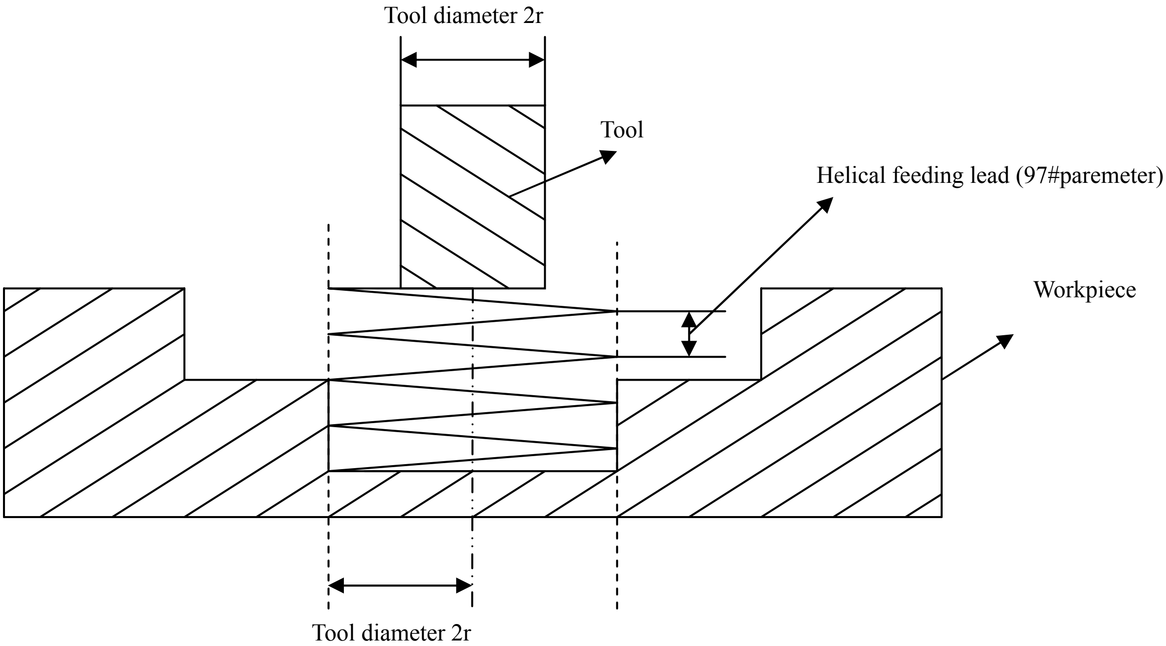
If setting value is more than or equal to 10, it feeds by helical type for rough milling command G110, G111, G134, G135.

Rough milling command（G110,G111,134,G135）helical feed function:

Namely, for Z axis depth cutting of rough milling command G110, G111, 134, G135, the tool feeds not by linear type, but by helical type. So the workpiece with no groove may be rough milled directedly.

Note 1 when the Z axis cutting depth is less than 10μm each time, the helical feeding is invalid.
Note 2 when the tool radius is less than 1mm, the helical feeding is also invalid.

The helical feeding path is shown in follows:



Volume III Installation

189	Movement per rotation of the 4th axis
190	Movement per rotation of the 5th axis

Setting range：1～9999999（unit：0.001deg）

201	Allowded valid ey number at the same time

Setting range：2～5

202	Define the name of the 4th axis(A:65, B:66, C:67)
203	Define the name of the 5th axis(A:65, B:66, C:67)

Setting range：65～67　65-A，66-B，67-C

213	Total tool number selection

Setting range：1～32

214	Reset output time

Setting range：16～4080（unit：ms）

215	Serial communication baudrate

Setting range：1200, 2400, 4800, 9600, 19200, 38400, 57600, 115200（unit：bit/s）

216	Block No. increment for block No.auto insertion

Setting range：1～100

3.2 Parameter description (by function sequence)

3.2.1 Axis control logic

008	DISP	***	***	**DIR5**	**DIR4**	**DIRZ**	**DIRY**	**DIRX**

DIR5 =1：Direction signal (DIR)is high level as the 5th axis moves positively;
=0：Direction signal (DIR)is low level as the 5th axis moves negatively.
DIR4 =1：Direction signal (DIR)is high level as the 4th axis moves positively;
=0：Direction signal (DIR)is low level as the 4th axis moves negatively.
DIRZ =1：Direction signal (DIR)is high level as Z axis moves positively;
=0：Direction signal (DIR)is low level as Z axis moves negatively.
DIRY =1：Direction signal (DIR)is high level as Y axis moves positively;
=0：Direction signal (DIR)is low level as Y axis moves negatively.
DIRX =1：Direction signal (DIR)is high level as X axis moves positively;
=0：Direction signal (DIR)is low level as X axis moves negatively.

0	0	9

***	***	***	**ALM5**	**ALM4**	**ALMZ**	**ALMY**	**ALMX**

ALM5 =1：the 5th axis low level alarm signal (ALM5);
=0：the 5th axis high level alarm signal (ALM5).
ALM4 =1：the 4th axis low level alarm signal (ALM4);
=0：the 4th axis high level alarm signal (ALM4).
ALMZ =1：Z axis low level alarm signal (ALMZ);
=0：Z axis high level alarm signal (ALMZ).
ALMY =1：Y axis low level alarm signal (ALMY);
=0：Y axis high level alarm signal (ALMY).
ALMX =1：X axis low level alarm signal (ALMX);
=0：X axis high level alarm signal (ALMX).

0	1	9

KEY1	***	***	**HNG5**	**HNG4**	**HNGZ**	**HNGY**	**HNGX**

HNG5 =1：the 5th MPG:ccw:+,cw:-;
=0：the 5th MPG:ccw:-,cw:+.
HNG4 =1：the 4th MPG:ccw:+,cw:-;
=0：the 4th MPG:ccw:-,cw:+.
HNGZ =1：Z MPG:ccw:+,cw:-;
=0：Z MPG:ccw:-,cw:+.
HNGY =1：Y MPG:ccw:+,cw:-;
=0：Y MPG:ccw:-,cw:+.
HNGX =1：X MPG:ccw:+,cw:-;
=0：X MPG:ccw:-,cw:+.

0	2	0

SPFD	SAR	THDA	**VAL5**	**VAL4**	**VALZ**	**VALY**	**VALX**

VAL5 =1：For the 5th axis move key,↑ is positive，↓is negative;
=0：For the 5th axis move key, ↓is positive，↑is negative.
VAL4 =1：For the 4th axis move key,↑ is positive，↓is negative;
=0：For the 4th axis move key, ↓is positive，↑is negative.
VALZ =1：For Z axis move key,↑ is positive，↓is negative;
=0：For Z axis move key, ↓is positive，↑is negative.
VALY =1：For Y axis move key,↑ is positive，↓is negative;
=0：For Y axis move key, ↓is positive，↑is negative.
VALX =1：For X axis move key, →is positive，←is negative;
=0：For X axis move key, ←is positive，→is negative

0	4	9		CMRX：X axis multiplier coefficient
0	5	0		CMRY：Y axis multiplier coefficient
0	5	1		CMRZ：Z axis multiplier coefficient
0	5	2		CMR4：4th axis multiplier coefficient
0	5	3		CMR5：5th axis multiplier coefficient

Setting range: 1～32767

0	5	4		CMDX：X axis frequency division coefficient
0	5	5		CMDY：Y axis frequency division coefficient
0	5	6		CMDZ：Z axis frequency division coefficient
0	5	7		CMD4：4th axis frequency division coefficient
0	5	8		CMD5：5th axis frequency division coefficient

Setting range： 1～32767

Electronic gear ratio formula： $\frac{CMR}{CMD}=\frac{S\times 360}{\alpha\times L}\times\frac{Z_M}{Z_D}$

S：Min. command output unit

Z_M：belt wheel teeth of lead screw

α: motor rotation angle for a pulse

Z_D：Wheel teeth of motor belt

L：Screw lead

3.2.2 Acceleration & deceleration control

0	0	4		***	**RDRN**	DECI	***	PROD	***	***	SCW

RDRN =1：G00 rapid traverse， speed = federate ×dry run speed;

=0：G00 speed = rapid override × rapid tranverse speed .

0	1	2		***	***	***	TMANL	***	***	EBCL	**ISOT**

ISOT =1：Prior to machine zero return after power on, manual rapid traverse valid;

=0：Prior to machine zero return after power on, manual rapid traverse invalid.

0	5	9		X axis max. rapid traverse speed
0	6	0		Y axis max. rapid traverse speed
0	6	1		Z axis max. rapid traverse speed
0	6	2		4th axis max. rapid traverse speed
0	6	3		5th axis max. rapid traverse speed

Setting range:10～1843143750（unit：mm/min）

Volume III Installation

0	6	4	Acceleration&deceleration time constant of X axis rapid traverse (ms)
0	6	5	Acceleration&deceleration time constant of Y axis rapid traverse (ms)
0	6	6	Acceleration&deceleration time constant of Z axis rapid traverse (ms)
0	6	7	Acceleration&deceleration time constant of 4th axis rapid traverse (ms)
0	6	8	Acceleration&deceleration time constant of 5th axis rapid traverse (ms)

Setting range:10～4000(unit：ms)

0	6	9	Rapid traverse speed when rapid override is F0

Setting range:6～4000（unit：mm/min）

0	7	0	Axes top feedrate of cutting

Setting range:10～15000（unit:mm/min）

0	7	1	Exponential acceleration start speed and deceleration end speed in cutting feed

Setting range:0～8000（unit:mm/min）

0	7	2	Exponential acceleration&deceleration time constant of cutting

Setting range:10～4000（unit：ms）

0	7	3	Start speed in manual feed.

Setting range:0～8000（unit:mm/min）

0	7	4	Exponential acceleration&deceleration time constant of manual feed

Setting range:10～4000（unit：ms）

3.2.3 Machine protection

0	1	7	***	**MST**	**MSP**	**MOT**	**MESP**	***	***	***

MST =1：External cycle start signal (ST) invalid，

=0：External cycle start signal (ST) valid.

MSP =1：External stop signal (SP) invalid，

=0：External stop signal (SP) valid with external stop switch connected, otherwise CNC shows "stop" .

MOT =1：Not detect software stroke limit;

=0：Detect software stroke limit.

MESP =1：Emergency stop invalid;

=0：Emergency stop valid

0	1	8	***	***	***	**ESCD**	***	***	***	***

ESCD =1：S code off at emergency stop;

=0：S code not off at emergency stop

Volume Ⅲ Installation

0	2	2		CALH	**SOT**	***	MZR5	MZR4	MZRZ	MZRY	MZRX

SOT =1：Software limit valid after zero return at power on;

=0：Software limit valid after power on.

1	3	5	Max. X coordinate value of software limit
1	3	6	Max. Y coordinate value of software limit
1	3	7	Max. Z coordinate value of software limit
1	3	8	Max. 4th coordinate value of software limit
1	3	9	Max. 5th coordinate value of software limit
1	4	0	Min. X coordinate value of software limit
1	4	1	Min. Y coordinate value of software limit
1	4	2	Min.Z coordinate value of software limit
1	4	3	Min. 4th coordinate value of software limit
1	4	4	Min. 5th coordinate value of software limit

Setting range：-9999999～+9999999（unit：0.001mm）

3.2.4 Thread function

0	2	0		SPFD	SAR	**THDA**	VAL5	VAL4	VALZ	VALY	VALX

THDA =1：Threading machining adopts exponential acceleration and deceleration;

=0：Threading machining adopts linear acceleration and deceleration.

0	7	5	Threading axes start speed

Setting range：6～8000（unit:mm/min）

3.2.5 Spindle control

0	0	1		***	***	***	**ACS**	HWL	***	***	***

ACS =1: Analog voltage control of spindle speed;

=0: Switching control of spindle speed.

0	9	9	Voltage compensation for 0V analog voltage output

Setting range：-1000～1000 （unit:mV）

1	0	0	Voltage offset value when spindle max. speed analog voltage 10V output

Setting range：-2000～2000（unit：mV）

Volume III Installation

1	0	1		Max spindle speed of 1st gear when analog voltage output is 10V
1	0	2		Max.spindle speed of 2nd gear when analog voltage output is 10V
1	0	3		Max.spindle speed of 3rd gear when analog voltage output is 10V
1	0	4		Max.spindle speed of 4th gear when analog voltage output is 10V

Setting range：10～9999 （unit:r/min）

1	0	7		Delay of spindle speed in-position signal detection

Setting range：0～4080 （unit:ms）

1	0	8		Max. spindle speed fluctuation allowed by system

Setting range：50～1000（unit:r/min）

1	0	9		spindle encoder pulses/rev

Setting range：0～5000 （unit：p/r）0: Not detect spindle encoder in G74, G84 tapping.

1	1	0		Transmission ratio of encoder and - spindle gear teeth
1	1	1		Transmission ratio of encoder and - encoder gear teeth

Setting range:1～255

3.2.6 Tool function

0	0	2		***	***	***	**LIFJ**	**MDITL**	**LIFC**	**NRC**	**TLIF**

LIFJ =1: Tool life management group skip valid;

=0: Tool life management group skip invalid.

MDITL =1: Tool life management valid in MDI mode;

=0: Tool life management invalid in MDI mode.

LIFC =1: Tool life counting type 2 by times;

=0: Tool life counting type 1 by times.

NRC =1: Tool nose radius compensation valid;

=0: Tool nose radius compensation invalid.

TLIF =1: Tool life management valid;

=0: Tool life management invalid

0	1	2		***	***	***	**TMANL**	***	***	EBCL	ISOT

TMANL =1：Manual tool change for T code;

=0：Auto tool change for T code.

2	1	3		Total tool number selection

Setting range：1～32

3.2.7 Edit and Display

0	0	4

***	RDRN	DECI	***	PROD	***	***	SCW

PROD =1：Relative coordinate displayed in POSITION page is programming position;

=0：Relative coordinate displayed in POSITION page is position involving tool offset.

0	0	8

DISP	***	***	DIR5	DIR4	DIRZ	DIRY	DIRX

DISP =1：Enter absolute page after power on;

=0：Enter relative page after power on.

0	1	2

***	***	***	TMANL	***	***	EBCL	ISOT

EBCL =1：Program end sign EOB displays “;”(semicolon);

=0：Program end sign EOB displays “*”(asterisk).

0	4	0

***	***	***	***	***	L2	L1	L0

L2, L1, L0：Interface language selection;

Language	L2	L1	L0
Chinese	0	0	0
English	0	0	1
Frence	0	1	0
Spanish	0	1	1
Germen	1	0	0
Italy	1	0	1
Russian	1	1	0
Korean	1	1	1

Volume III Installation

2	1	6

Block No. increment for block No.auto insertion

Setting range：1～100

3.2.8 Precision compensation

0	0	3

***	***	PCOMP	***	***	***	D/R	***

PCOMP =1: Screw-pitch error compensation valid;

=0: Screw-pitch error compensation invalid.

D/R =1: Tool offset D value is diameter input;

=0: Tool offset D value is radius input.

0	1	0

CPF7	CPF6	CPF5	CPF4	CPF3	CPF2	CPF1	CPF0

CPF0～CPF7： Setting values of backlash compensation pulse frequency.

The set frequency =

（2^7×CPF7+2^6×CPF6+2^5×CPF5+2^4×CPF4+2^3×CPF3+2^2×CPF2+2^1×CPF1+CPF0）Kpps

0 1 1	BDEC	BD8	***	***	***	ZNIK	***	***

BDEC =1：Backlash compensation type B, the compensation data are output by ascending or decending type and the set frequency is invalid.;

=0：Backlash compensation type A, the compensation data are output by the set frequency (set by bit parameter No.010) or 1/8 of it.

BD8 =1：Backlash compensation is done by the 1/8 of the set frequency;

=0：Backlash compensation is done by the set frequency.

0 2 2	CALH	SOT	***	MZR5	MZR4	MZRZ	MZRY	MZRX

CALH =1：Length offset not cancel in reference point return;

=0：Length offset cancel in reference point return.

115	X axis backlash offset
116	Y axis backlash offset
117	Z axis backlash offset
118	4th axis backlash offset
119	5th axis backlash offset

Setting range：0～2000（unit:0.001mm）

120	Interval of X axis screw-pitch error compensation
121	Interval of Y axis screw-pitch error compensation
122	Interval of Z axis screw-pitch error compensation
123	Interval of 4th axis screw-pitch error compensation
124	Interval of 5th axis screw-pitch error compensation

Setting range： 1000～999999（unit：0.001mm ）

125	Screw-pitch error compensation number of X axis machine zero
126	Screw-pitch error compensation number of Y axis machine zero
127	Screw-pitch error compensation number of Z axis machine zero
128	Screw-pitch error compensation number of the 4th axis machine zero
129	Screw-pitch error compensation number of the 5th axis machine zero

Setting range： 0～255

3.2.9 Communication setting

215	Serial communication baudrate

Setting range：1200, 2400, 4800, 9600, 19200, 38400, 57600, 115200 (unit:bit/s)

3.2.10 Machine zero return

0 0 4	***	RDRN	**DECI**	***	PROD	***	***	SCW

DECI =1：Deceleration signal high level for machine zero return;
=0：Deceleration signal low level for machine zero return.

0 1 1	BDEC	BD8	***	***	***	**ZNIK**	***	***

ZNIK =1：Direction keys locked during zero return, homing continues to end by pressing direction key once;
=0：Direction keys unlocked but should be held on during zero return

0 0 6	***	***	***	**ZM5**	**ZM4**	**ZMZ**	**ZMY**	**ZMX**

ZM5 =1：5th zero return type C;
=0：5th zero return type B.
ZM4 =1：4th zero return type C;
=0：4th zero return type B.
ZMZ =1：Z zero return type C;
=0：Z zero return type B.
ZMY =1：Y zero return type C;
=0：Y zero return type B.
ZMX =1：X zero return type C;
=0：X zero return type B.

0 0 7	AVGL	***	SMZ	**ZC5**	**ZC4**	**ZCZ**	**ZCY**	**ZCX**

ZC5 =1：The deceleration signal (DEC5) and one-rotation signal (PC5) of 5th axis in parallel connection (a proximity switch acting as both the deceleration signal and zero signal) during machine zero return;
=0：The deceleration signal (DEC5) and one-rotation signal (PC5) of 5th axis are connected independently (the indepent deceleration signal and zero signal are required) during machine zero return.

ZC4 =1：The deceleration signal (DEC4) and one-rotation signal (PC4) of 4th axis in parallel connection (a proximity switch acting as both the deceleration signal and zero signal) during machine zero return;
=0：The deceleration signal (DEC4) and one-rotation signal (PC4) of 4th axis are connected independently (the indepent deceleration signal and zero signal are required) during machine zero return.

ZCZ =1：The deceleration signal (DECZ) and one-rotation signal (PCZ) of Z axis in parallel connection (a proximity switch acting as both the deceleration signal and zero signal)

during machine zero return;

=0：The deceleration signal DECZ) and one-rotation signal (PCZ) of Z axis are connected independently (the indepent deceleration signal and zero signal are required) during machine zero return.

ZCY =1：The deceleration signal (DECY) and one-rotation signal (PCY) of Y axis in parallel connection (a proximity switch acting as both the deceleration signal and zero signal) during machine zero return;

=0：The deceleration signal (DECY)and one-rotation signal PCY) of Y axis are connected independently (the indepent deceleration signal and zero signal are required) during machine zero return.

ZCX =1：The deceleration signal (DECX) and one-rotation signal (PCX) of X axis in parallel connection (a proximity switch acting as both the deceleration signal and zero signal) during machine zero return;

=0：The deceleration signal (DECX) and one-rotation signal (PCX) of X axis are connected independently (the indepent deceleration signal and zero signal are required) during machine zero return.

0	1	4

***	***	***	ZRS5	ZRS4	ZRSZ	ZRSY	ZRSX

ZRS5 =1: There are machine zero point in the 5th axis, it detects deceleration signal and zero signal when performing machine zero return;

=0: There are no machine zero point in the 5th axis, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return.

ZRS4 =1: There are machine zero point in the 4th axis, it detects deceleration signal and zero signal when performing machine zero return;

=0: There are no machine zero point in the 4th axis, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return.

ZRSZ =1: There are machine zero point in Z axis, it detects deceleration signal and zero signal when performing machine zero return;

=0: There are no machine zero point in Z axis, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return.

ZRSY =1: There are machine zero point in Y axis, it detects deceleration signal and zero signal when performing machine zero return;

=0: There are no machine zero point in Y axis, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return.

ZRSX =1: There are machine zero point in X axis, it detects deceleration signal and zero signal when performing machine zero return;

=0: There are no machine zero point in X axis, it returns to machine zero without detecting deceleration signal and zero signal when performing machine zero return.

0	2	2		CALH	SOT	***	MZR5	MZR4	MZRZ	MZRY	MZRX

CALH =1：Length offset not cancel in reference point return;

=0：Length offset cancel in reference point return.

MZR5 =1：Machine zero return in negative the 5th axis;

=0：Machine zero return in positive the 5th axis.

MZR4 =1：Machine zero return in negative the 4th axis;

=0：Machine zero return in positive the 4th axis.

MZRZ =1：Machine zero return in negative Z axis;

=0：Machine zero return in positive Z axis.

MZRY =1：Machine zero return in negative Y axis;

=0：Machine zero return in positive Y axis.

MZRX =1：Machine zero return in positive X axis;

=0：Machine zero return in negative X axis.

089	Low speed of X axis machine zero return
090	Low speed of Y axis machine zero return
091	Low speed of Z axis machine zero return
092	Low speed of the 4th axis machine zero return
093	Low speed of the 5th axis machine zero return

Setting range：10～1000（unit：mm/min）

094	High speed of X axis machine zero return
095	High speed of Y axis machine zero return
096	High speed of Z axis machine zero return
097	High speed of the 4th axis machine zero return
098	High speed of the 5th axis machine zero return

Setting range：10～921571875 (unit:mm/min)

130	X axis machine zero offset
131	Y axis machine zero offset
132	Z axis machine zero offset
133	The 4th axis machine zero offset
134	The 5th axis machine zero offset

Setting range：-99999～99999(unit：0.001mm)

Volume III Installation

1	4	5	X machine coordinate of the 1st reference point
1	4	6	Y machine coordinate of the 1st reference point
1	4	7	Z machine coordinate of the 1st reference point
1	4	8	The 4th machine coordinate of the 1st reference point
1	4	9	The 5th machine coordinate of the 1st reference point
1	5	0	X machine coordinate of the 2nd reference point
1	5	1	Y machine coordinate of the 2nd reference point
1	5	2	Z machine coordinate of the 2nd reference point
1	5	3	The 4th machine coordinate of the 2nd reference point
1	5	4	The 5th machine coordinate of the 2nd reference point
1	5	5	X machine coordinate of the 3rd reference point
1	5	6	Y machine coordinate of the 3rd reference point
1	5	7	Z machine coordinate of the 3rd reference point
1	5	8	The 4th machine coordinate of the 3rd reference point
1	5	9	The 5th machine coordinate of the 3rd reference point
1	6	0	X machine coordinate of the 4th reference point
1	6	1	Y machine coordinate of the 4th reference point
1	6	2	Z machine coordinate of the 4th reference point
1	6	3	The 4th machine coordinate of the 4th reference point
1	6	4	The 5th machine coordinate of the 4th reference point

Setting range：-99999999～99999999 (unit:0.001mm)

3.2.11 Rotary axis function

0	2	5	**RTORI**	***	**RTPCP**	***	***	**RTCRG**	***	***

RTORI =1：M29 is executed,Spindle need to return zero;

=0：M29 is executed,Spindle need not to return zero.

RTPCP =1：Rigid tapping is the high-speed deep hole cycle(G73);

=0：Rigid tapping is the high-speed deep hole cycle (G83).

RTCRG =1：Do not wait for G61.0 to be 1 as excuting next program block after rigid tapping cancelled;

=0：Do wait for G61.0 to be 1 as excuting next program block after rigid tapping cancelled.

0	2	6	***	***	***	**RCS4**	***	***	**ROS4**	**ROT4**

RCS4 =1：Cs function of 4th axis is valid(power on);

=0：Cs function of 4th axis is invalid(power on).

ROS4, ROT4：Set the type of 4th axis;

	Linear	Rotary A	Rotary B	invalid
ROT4	0	1	1	0
ROS4	0	0	1	1

0	2	7		***	RRT4	***	***	***	RRL4	RAB4	ROA4

RRT4 =1：Zero mode D is used on the 4th rotary axis (power on);

=0：Zero mode A,B,C are used on the 4th rotary axis (power on).

RRL4 =1：the 4th rel.coor.cycle func.is valid (power on);

=0：the 4th rel.coor.cycle func.is invalid(power on).

RAB4 =1：the 4th rotates according to symbol direction;

=0：the 4th rotates according to nearby rotation.

ROA4 =1：the 4th abs.coor.cycle func.is valid (power on);

=0：the 4th abs.coor.cycle func.is invalid(power on).

0	2	8		***	***	***	RCS5	***	***	ROS5	ROT5

RCS5 =1：Cs function of the 5th axis is valid(power on);

=0：Cs function of the 5th axis is invalid(power on).

ROS5, ROT5：Set the type of 5th;

	Linear	Rotary A	Rotary B	invalid
ROT5	0	1	1	0
ROS5	0	0	1	1

0	2	9		***	RRT5	***	***	***	RRL5	RAB5	ROA5

RRT5 =1：Zero mode D of the 5th axis (power on) ;

=0：Zero mode A, B, C of the 5th axis (power on) .

RRL5 =1：the 5th rel.coor.cycle func.is valid (power on);

=0：the 5th rel.coor.cycle func.is invalid(power on).

RAB5 =1：the 5th rotation according to symbol direction;

=0：the 5th rotation according to nearby direction.

ROA5 =1：the 5th abs.coor.cycle func.is valid (power on);

=0：the 5th abs.coor.cycle func.is invalid(power on).

RRT4 =1：Zero mode D is used on the 5th rotary axis (power on);

=0：Zero mode A,B,C are used on the 5th rotary axis (power on).

RRL4 =1：the 5th rel.coor.cycle func.is valid (power on);

=0：the 5th rel.coor.cycle func.is invalid(power on).

RAB4 =1：5th rotates according to symbol direction;

=0：5th rotates according to nearby rotation.

ROA4 =1：the 5th abs.coor.cycle func.is valid (power on);

=0：the 5th abs.coor.cycle func.is invalid(power on).

0	7	7	Initial speed of acc.&dec in using CS funciton

Setting range： 0～5000（Unit:deg/min）

0	7	8	Acc.&dec.time constant in using CS function

Setting range： 10～10000（Unit:ms）

0	8	1	Initial speed of linear acceleration/deceleration in rigid tapping

Setting range： 0～5000（Unit:mm/min）

0	8	2	Linear time constant in rigid tapping tool infeed

Setting range： 10～10000（Unit:ms）

0	8	3	Time constant in rigid tapping tool retract

Setting range： 0～4000（Unit:ms）, 082 setting value is used when it is set to 0.

0	8	4	Override value in rigid tapping tool retract(0: override is set to 100%)

Setting range： 0～200, 0: override is set to 100%

0	8	5	Tool retract amount in deep hole rigid tapping(high-speed, standard)

Setting range：0～32767000,（Unit:0.001mm）

1	8	9	One-rotaton increment of the 4th axis
1	9	0	One-rotaton increment of 5th axis

Setting range：1～9999999,（Unit:0.001deg）

2	0	1	Amount of valid keys pressed simultaneously

Setting range：2～5

2	0	2	Define the name of the 4th axis (A:65, B:66, C:67)
2	0	3	Define the name of the 5th axis (A:65, B:66, C:67)

Setting range：65～67 65-A，66-B，67-C

CHAPTER 4 MACHINE DEBUGGING METHODS AND STEPS

The trial run methods and steps at initial power on for this GSK980MDa are described in this chapter. The corresponding operation can be performed after the debugging by the following steps.

4.1 Emergency Stop and Stroke Limit

This GSK980MDa system has software limit function, it is suggested that the stroke limit switches are fixed in the positive or negative axes for hardware limit. The connection is shown in follows:（The chart is designed for X, Y, Z axes）

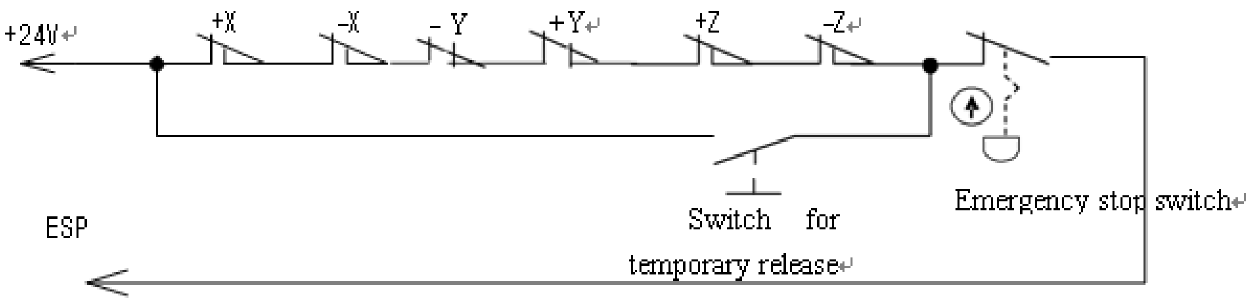


Fig.4-1

So the MESP of bit parameter No.17should be set to 0.

And the CNC diagnostic message ESP can monitor the state of emergency stop input signal.

In Manual or MPG mode, slowly move the axes to test the validity of stroke limit switch, correctness of alarm display, validity of overtravel release button.When the overtravel occurs or Emergency Stop button is pressed,"emergency stop" alarm will be issued by CNC system. The alarm can be cancelled by pressing down the Overtravel button and moving reversely.

4.2 Drive unit Unit Setting

Set BIT4～BIT0 of bit parameter No.009 according to alarm logic level of drive unit. The BIT4～BIT0 of bit parameter No.009 for our drive unit are all set for 1 .

If the machine moving direction is not consistent with the moving command, modify the BIT4～BIT0 of bit parameter No.008，BIT4～BIT0 of bit parameter No.019, BIT4～BIT0 of bit parameter No.20.

4.3 Gear Ratio Adjustment

The data parameter No.049～No.058 can be modified for electronic gear ratio adjustment to meet the different mechanical transmission ratio if the machine travel distance is not consistent with the displacement distance displayed by the CNC coordinate.

Calculation formula:

$$\frac{CMR}{CMD}=\frac{\delta\times 360}{\alpha\times L}\times\frac{Z_M}{Z_D}$$

CMR: command multiplier coefficient (data parameter №049, №050, №051, №052, №053)

CMD: command frequency division coefficient (data parameter №054, №055, №056, №057, №058)

α :: pulse volume, motor rotation angle for a pulse

L: lead

δ: min. input command unit of CNC (0.0001 for all axes of GSK980MDa)

ZM: gear teeth of lead screw

ZD: gear teeth of motor

If the electronic gear ratio numerator is greater than the denominator, the allowed CNC max. speed will decrease. For example: the data parameter No.051 (CMRZ) =2 , №056 (CMDZ) =1, so the allowed Z axis max. speed is 8000mm/min.

If the electronic gear ratio numerator is not equal to the denominator, the allowed CNC positioning precision will decrease. For example: when the data parameter No.051 (CMRZ)=1 and №056 (CMDZ)=5, the pulse is not output as the input increment is 0.004, but a pulse is output if the input increment is up to 0.005.

In order to ensure the CNC positioning precision, speed index and match with digit servo with electronic gear ratio function, it is suggested that the CNC electronic gear ratio is set for 1:1 or the electronic gear ratio calculated is set to the digital servo.

When matching with the step drive, choose the drive unit with step division function as far as possible, and properly select mechanical transmission ratio. The 1:1 electronic gear ratio should be ensured to avoid the too large difference between the numerator and the denominator of this CNC gear ratio.

Example:

Match GSK980MDa with DA98B, take X axis for example: set command multiplier coefficient and command frequency division coefficient to 1. Calculation formula is shown below.

CNC：

$$\frac{CMR}{CMD}=\frac{\delta\times 360}{\alpha\times L}\times\frac{Z_M}{Z_D}=\frac{1}{1}$$

The following conclusions can be reached:

$$\alpha=\frac{\delta\times 360}{L}\times\frac{Z_M}{Z_D}\ (\text{deg/pulse})$$

Drive unit：

Parameters 12, 13 of drive unit correspond to position command pulse frequency division

molecule and denominator. Calculation formula of drive unit gear ratio is shown as follows:

$$P \times G = 4 \times N \times C$$

P：Correspondence between required pulse volume for motor rotates 3600 and CNC end： $P = 360/\alpha$

G： Electronic gear ratio of drive unit, G= position command pulse frequency division molecule/ position command pulse frequency division denominator

N： Set motor rev number to 1

C： Wire number of feedback encoder: DA98B is 2500p/r.

The following conclusions can be reached:

$$G = \frac{4 \times N \times C}{P} = 4 \times N \times C \times \frac{\alpha}{360} = \frac{4 \times N \times C}{360} \times \frac{\delta \times 360}{L} \times \frac{Z_M}{Z_D} =$$

$$= \frac{10 \times Z_M}{L \times Z_D}$$

Set molecule and denominator of caculated ratio to drive unit 12, 13 separately.

4.4 Acceleration&deceleration Characteristic Adjustment

Adjust the relative CNC parameters according to the factors such as the drive unit, motor characteristics and machine load:

Data parameter №059～№063：X, Y, Z, 4th, 5th axis rapid traverse rate;

Data parameter №064～№068: linear acceleration & deceleration time constant of X, Y, Z, 4th, 5th axis rapid traverse rate;

Data parameter №069: rapid traverse speed when rapid override is F0

Data parameter №070: upper limit of axes cutting feedrate;

Data parameter №071: Start/end speed of exponential acceleration & deceleration in cutting feeding;

Data parameter №072: Exponential acceleration & deceleration time constant of cutting feeding;

Data parameter№073：Start/end speed of exponential acceleration & deceleration in MPG/Step feedrate;

Data parameter№074 ： Exponential acceleration & deceleration time constant of MPG/STEP/manual feed;

Data parameter№075：Start/end speed in thread cutting of each ax;

Data parameter№077：Initial feedrate of acc.&dec in CS axis;

Data parameter№078：Acc.&dec.time constant in CS axis;

Data parameter№081：Initial speed of linear acceleration/deceleration in rigid tapping;

Data parameter№082：Linear acceleration/deceleration time constant in rigid tapping tool infeed;

Data parameter№083：Linear acceleration/deceleration time constant in rigid tapping tool retraction;

Data parameter№084：Override value in rigid tapping tool retract;

Data parameter№172：Initial feedrate when power on;

Data parameter№174：Feedrate of DRY run;

SMZ of bit parameter №007: for validity of smoothing transition between blocks

The larger the acceleration&deceleration time constant is, the slower tacceleration&deceleration is, the smaller the machine movement impact and the lower the machining efficiency is.And vice versa.

If acceleration&deceleration time constants are equal, the higher the acceleration & deceleration start/end speed is, the faster the acceleration & deceleration is, the bigger the machine movement impact and the higher the machining efficiency is. And vice versa.

The principle for acceleration&deceleration characteristic adjustment is to properly reduce the acceleration & deceleration time constant and increase the acceleration&deceleration start/end speed to improve the machining efficiency on the condition that there is no alarm, motor out-of-step and obvious machine impact. If the acceleration&deceleration time constant is set too small, and the start/end speed is set too large, it is easily to cause drive unit alarm, motor out-of-step or machine vibration.

When the bit parameter №007 BIT3（SMZ）=1, the feedrate drops to the start speed of the acceleration&deceleration at the cutting path intersection, then it accelerates to the specified speed of the adjacent block to obtain an accurate positioning at the path intersection, but this will reduce the machining efficiency. When SMZ=0, the adjacent cutting path transits smoothly by the acceleration&deceleration. The feedrate does not always drop to the start speed when the previous path is finished and a circular transition (non-accurate positioning) will be formed at the path intersection. The machining surface by this path transition has a good finish and a higher machining efficiency. When the stepper motor drive unit is applied, the SMZ of the bit parameter №007 should be set to 1 to avoid the out-of-step.

When the stepper motor drive unit is applied to this system, the out-of-step may occur if rapid traverse speed is too large, acceleration&deceleration time constant is too small, acceleration&deceleration start/end speed is too large. The suggested parameter setting is shown in follows (the electronic gear ratio is 1:1):

Data parameter №059~№063≤5000　Data parameter №064~№068≥350　Data parameter №071≤50

Data parameter №072≥150　Data parameter №073≤50　Data parameter№074≥150

Data parameter№075≤100

When AC servo motor drive unit is applied to this system, the machining efficiency can be improved by a larger start speed and smaller ACC&DEC time constant setting. If optimum ACC&DEC characteristics are required, the ACC&DEC time constant may be set to 0，which can be got by adjusting the AC servo ACC&DEC parameters. The suggested parameter settings are as follows (electronic gear ratio is 1:1).

Data parameter №059~№063　set higher properly

Data parameter №064~№068≤60

Data parameter №071≥50

Data parameter №072≤50

Data parameter №073≥50

Data parameter №074≤50

Data parameter №075≤500

The parameter settings above are recommended for use, refer to the actual conditions of the drive unit, motor characteristic and machine load for its proper setting.

4.5 Machine Zero Adjustment

Adjust the relevant parameters based on the valid level of the connection signal, zero return type or direction applied:

（DECI）of the bit parameter №004: valid level of deceleration signal as machine zero return

(ZM5~ZMX) of the bit parameter №006: return and initial backlash direction of X, Y, Z，4th, 5th axes machine zeroes at deceleration.

(ZC5~ZCX) of the bit parameter №007: it is able to set whether an approach switch taken as both deceleration and zero signals when X, Y, Z, 4th, 5^{th} axes return to machine zero point.

（ZNLK）of the bit parameter №011: for direction keys lock when performing zero return

(ZRS5~ZRSX) of the bit parameter №014: for deceleration and zero signals detection of X, Y, Z axes in machine zero return.

(MZR5~MZRX）of the bit parameter №22: for positive or negative zero turn of X, Y, Z, 4th, 5th axes

Data parameter №089~№093: low speed of X, Y, Z, 4th, 5th axes in machine zero return

Data parameter №094~№098: high speed of X, Y, Z, 4th, 5th axes in machine zero return

RRT4 of bit parameter №027 and RRT5 of №029 set the machine zero return type of the 4th and the 5th axis separately.

Machine zero return can be done after the validity of overtravel limit swithch is confirmed.Machine zero return types A, B, C can be selected for basic axes (X, Y, Z). Machine zero return types A, B, C, D can be selected for additional axes (4th, 5th).

The machine zero is usually fixed at the max. travel point, and the effective stoke of the zero return touch block should be more than 25mm to ensure a sufficient deceleration distance for accurate zero return. The more rapid the machine zero return is, the longer the zero return touch block should be. Or the moving carriage will rush through the block which may influence the zero return precision because of the insufficient deceleration distance.

Usually there are 2 types of machine zero return connection:

1 The connection to AC servo motor: schematic diagram of using a travel switch and a servo motor one-rotation signal separately

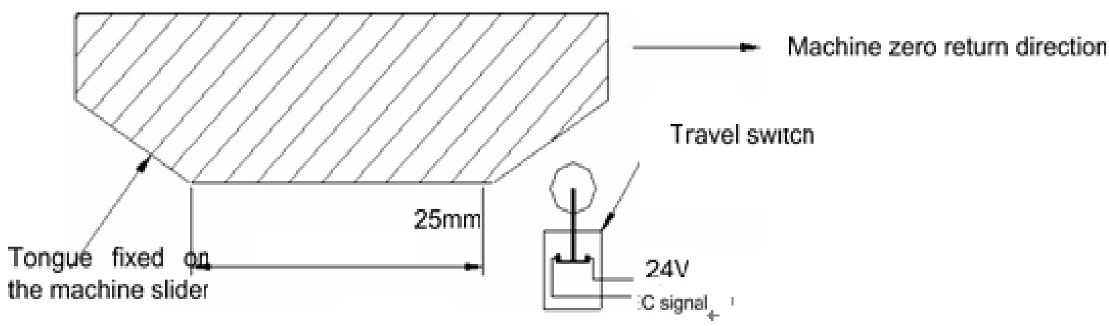


Fig. 4-2

By this connection type, when the deceleration switch is released in machine zero return, the one-rotation signal of encoder should be avoided to be at a critical point after the travel switch is released.In order to improve the zero return precision，it should be ensured the motor reaches the one-rotation signal of encoder after it rotates for half circle.And the moving distance for motor half circle rotation is the motor gear teeth/(2×lead screw gear teeth)

2 The connection to stepper motor: the schematic diagram of using a proximity switch taken as both deceleration signal and zero signal

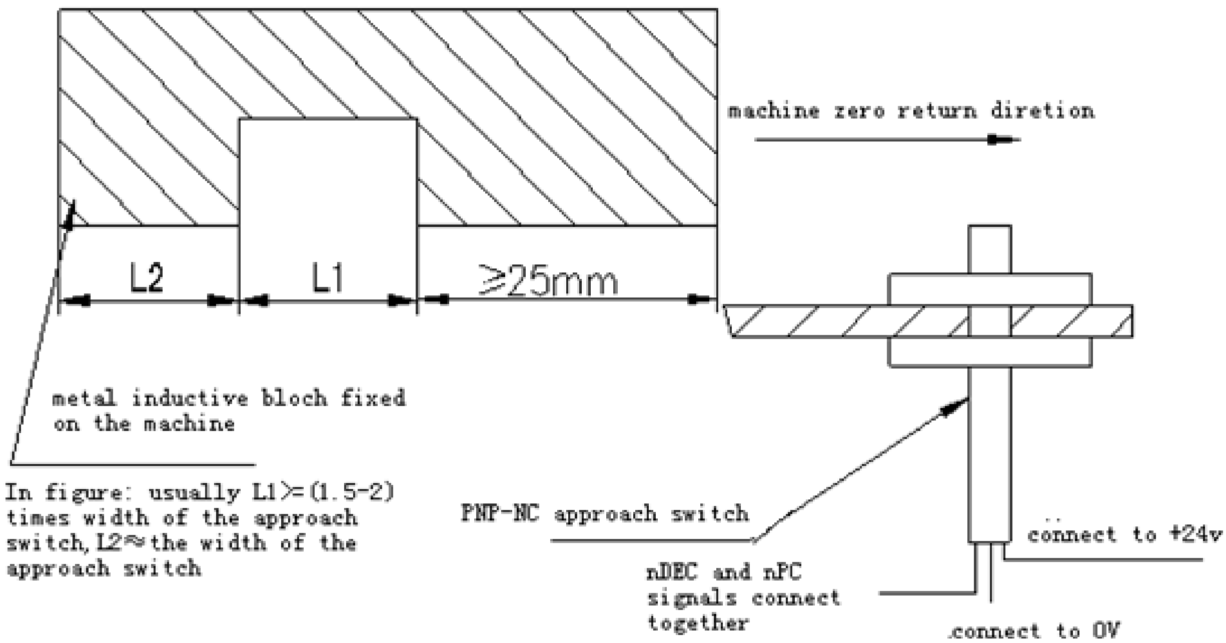


Fig 4-3

4.6 Spindle Adjustment

4.6.1 Spindle encoder

Encoder with the linear number 100~5000p/r is needed to be installed on the machine for threading. The linear number is set by data parameter No.109. The transmission ratio(spindle gear teeth/encoder gear teeth) between encoder and spindle is 1/255～255. The spindle gear teeth are set by CNC data parameter No.110, and the encoder gear teethare set by data parameter No.111. Synchronous belt transmission should be applied for it (no sliding transmission).

The DGN.011 and DNG.012 of CNC diagnosis messages are used to check the validity of threading signal from the spindle encoder.

4.6.2 Spindle brake

After spindle stop is executed, proper spindle brake time should be set to stop the spindle promptly in order to enhance the machining efficiency. If the brake is employed with energy consumption type, too long braking time may damage the motor. So the brake time is set by PLC.

4.6.3 Switch volume control of spindle speed

When multiple speed motor control is used, motor speed control command can be defined by ladder diagram as S_ _. Relevant parameter is shown below.

Bit parameter №001 ACS=0：select switching control of spindle speed.

4.6.4 Analog voltage control for spindle speed

This function can be obtained by the parameter setting of CNC. By interface outputting 0V～10V analog voltage to control inverter, the stepless shift can be obtained. And the related parameters are needed to be adjusted are:

Bit parameter №001 ACS=1 : for selection of spindle speed analog voltage control;

Data parameter №099: offset compensation value as spindle speed command voltage is 0V;

Data parameter №0100: offset compensation value as spindle speed command voltage is 10V;

Data parameter №101 ~ №104 :Max. speed limit for spindle speed gear 1 ~ 4. When CNC power on, the defaulted gear is 1 for spindle.

Basic parameters needed to be adjusted for inverter (refer to the relavant inverter manual for specific adjustment): CCW or CW command mode is selected by frenauency.

If the speed by programming is not consistent with that detected by the encoder, it can be adjusted to be consistent with the actual one by adjusting the data parameter №101～№104.

Speed adjustment method: select the spindle first gear, input S9999 code in MDI mode to run the spindle, view the spindle speed shown on the right bottom of the screen, then reinput the displayed speed value into the parameter №101. The other spinle gear adjustment is identical with this.

When entering S9999 code, the voltage should be 10V, S0 is 0V. If there is an voltage error, adjust bit parameter №099 and №100 to correct the voltage offset value(corrected by manufacturer, usually not needed).

When the current gear is the max.speed, if the analog voltage output by CNC is higher than 10V, set a smaller value for data parameter №100; when the S00 code is entered, if there is still slow rotation in the spindle, it means the analog voltage output by CNC is higher than 0V, so set a smaller value for data parameter №099.

If the machine is not fixed with an encoder, the spindle speed can be detected by a speed sensor, input S9999 in MDI mode to set the speed value displayed by sensor to the data parameter №101.

4.7 Backlash Offset

The backlash offset is input by diameter value with the unit 0.001mm, which is irrelevant to the programming by diameter or by radius. It can be measured by a dial indicator, a micrometer or a laser detector. Because the backlash offset can improve the machining precision only by accurate compensation, it is not recommended to measure it in MPG or Step mode, but the following method is suggested:

- Program editting

  O0001;
  N10 G01 Z10 F800 G91 ;
  N20 Z15 ;
  N30 Z1 ;
  N40 Z-1 ;
  N50 M30 .
- Set the backlash error offset to 0 before measuring:

- Run the program by single blocks, search the measuring benchmark A after 2 positioning operations, record the current data, move 1mm in the same direction, then move 1mm reversely to point B, read the current data.

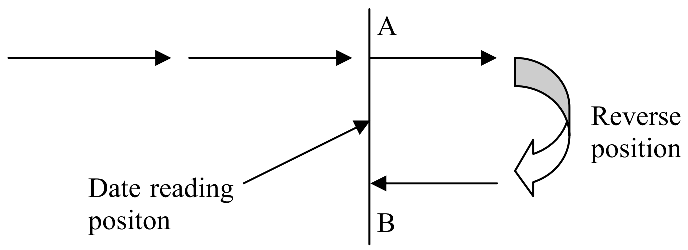


Fig. 4-4 Schematic map of backlash measuring methods

Backlash error offset value =| data of point A –data of point B |.Input the calculated data to the CNC data parameter №115～№119. Calculation for other axes are the same as this.

Data A ：dial-indicator data at point A

Data B ：dial-indicator data at point B

Note 1: The backlash offset mode and offset frequency can be set by BDEC and BD8 of bit parameter №011.

Note 2: Check the machine backlash at regular intervals according to specific conditions to ensure machine precison.

4.8 Step/MPG adjustment

The MPG key on the panel can be used to select the Step mode or MPG mode, which is set by the HWL of bit parameter №001.

HWL =1：MPG mode valid, Step mode invalid;

=0：Step mode valid, MPG mode invalid;

The dirtction 0f rotation for handwheel can be adjusted by parameter:

0	1	9

***	***	***	HNG5	HNG4	HNGZ	HNGY	HNGX

HNG5 =1：5th handwheel: ccw: +, cw:-;

=0：5th handwheel: ccw:-, cw: +.

Volume III Installation

HNG4 =1：4th handwheel: ccw: +, cw:-;

=0：4th handwheel: ccw:-, cw: +

HNGZ =1：Z handwheel: ccw: +, cw: -;

=0：Z handwheel: ccw: -, cw: +.

HNGY =1：Y handwheel: ccw: +,cw: -;

=0：Y handwheel: ccw: -, cw: +.

HNGX =1：X handwheel: ccw: +, cw: -;

=0：X handwheel: ccw: -,cw: +.

4.9 Other Adjustment

0	1	7		***	MST	MSP	MOT	MESP	***	***	***

MST =1： External Cycle Start (ST) signal invalid.

=0： External Cycle Start(ST) signal valid;

MSP =1： External Dwell (SP) signal invalid.

=0： External Stop (SP) signal valid.

MOT =1： Not check software limit.

=0： Check software limit;

MESP =1： External ESP signal invalid;

=0： External ESP signal valid.

0	1	8		***	***	***	ESCD	***	***	***	***

ESCD =1：S code off in emergency stop;

=0：S code not off in emergency stop

CHAPTER 5 DIAGNOSIS MESSAGE

5.1 CNC Diagnosis

This diagnosis section is used to check the CNC interface signals and internal running state and it can not be modified.

5.1.1 Signal diagnosis from machine to CNC

0 0 0								
	ESP			DEC5	DEC4	DECZ	DECY	DECX
Pin No.	CN61.6			CN61.34	CN61.33	CN61.12	CN61.32	CN61.4
PLC fixed address	X0.5			X2.5	X2.4	X1.3	X2.3	X0.3

XDEC, YDEC, ZDEC, DEC4, DEC5: Deceleration signal of X, Y, Z, 4th, 5th axes machine zero

ESP: Emergency signal

0 0 1								
								SKIP
Pin No.								CN61.42
PLC fixed address								X3.5

SKIP：Skip signal

5.1.2 Axes moving state and data diagnosis signal of CNC

0 0 3								
				RDY5	RDY4	RDYZ	RDYY	RDYX

RDYX~RDY5：The signal that(X, Y, Z, 4th, 5th) axis is ready

0 0 4								
	***	***	***	EN5	EN4	ENZ	ENY	ENX

ENX~EN5：The singnal that (X, Y, Z, 4th, 5th) axis is enabled

0 0 5								
	***	***	***	SET5	SET4	SETZ	SETY	SETX

SETX~SET5：

SETX~SET5：axis pulse prohibited signal

0 0 6								
	***	***	***	DRO5	DRO4	DROZ	DROY	DROX

DROX～DRO5：Output of (X, Y, Z, 4th, 5th) axis moving direction.

0 0 7								
	***	***	***	TDR5	TDR4	TDRZ	TDRY	TDRX

TDRX～TDR5：Direction of (X, Y, Z, 4th, 5th) axis moving path (1:positive; 0:negative)

0 0 8								
	***	***	***	PC5	PC4	PCZ	PCY	PCX

PCX～PC5：Zero point signal of (X, Y, Z, 4th, 5th) axis

Volume Ⅲ Installation

0	0	9	***	***	***	ALM5	ALM4	ALMZ	ALMY	ALMX

ALMX～ALM5：ALam signal of (X, Y, Z, 4th, 5th) axis

0	1	0	Handwheel speed data
0	1	1	Spindle feedback data
0	1	2	Spindle feedback data

0	1	3	Spindle analog voltage output
0	1	4	Spindle analog voltage output

5.1.3 MDI panel keys diagnosis

DGN.016～DGN.022 are the diagnosis messages of MDI keypad keys. When pressing a key in the operation panel, the corresponding bit displays "1" , and"0"after releasing this key. If it displays reversely, it means there is a fault in the keypad circuit.

0 1 6	RST	O	N	G	P/Q	7	8	9
Corresponding key	RESET	O	N	G	P Q	7	8	9

0 1 7	PGU	X	Y/&	Z/|	U/W	4	5	6
Corresponding key		X	Y &	Z |	U W	4	5	6

0 1 8	PGD	H	F/E	R/V	D/L	1	2	3
Corresponding key		H	F E	R V	D L	1	2	3

0 1 9	⇧	⇨	I/A	J/B	K/C	-/+/␣	0	./</>
Corresponding key	⇧	⇨	I A	J B	K C	- + ␣	0	. < >

0 2 0	⇩	⇦	M/[	S/]	T/=	EOB	ALT/MAC	DEL
Corresponding key	⇩	⇦	M [	S]	T =	EOB	OFFSET SETTING	DELETE

0 2 1		POS	PRG	OFT	ALM	SET	PAR	DGN
Corresponding key		POSITION	PROGRAM	OFFSET	ALARM	SETTING	PARAMETER	DIAGNOSIS

0	2	2	IN	OUT	CHG	//*/#	CAN			
Corresponding key			DATA INPUT	DATA OUTPUT	CHANGE	/ * #	CANCEL			

5.1.4 CNC internal state

During the CNC auto run, the current CNC running state can be viewed by DGN.064~DGN.110 diagnosis messages if there is no alarm and moving.

0	7	8	As power off,X start posion of executing segment
0	7	9	As power off,Y start posion of executing segment
0	8	0	As power off,Z start posion of executing segment
0	8	1	As power off,4th start posion of executing segment
0	8	2	As power off,5th start posion of executing segment

0	8	3	When the power off, G mode of group 01(G00～G03)
0	8	4	When the power off, G mode of group 02(G17～G19)
0	8	5	When the power off, G mode of group 03(G90, G91)
0	8	6	When the power off, G mode of group 05(G94, G95)
0	8	7	When the power off, G mode of group 06(G20, G21)
0	8	8	When the power off, G mode of group 07(G40～G42)
0	8	9	When the power off, G mode of group 08(G43/44/49)
0	9	0	When the power off, G mode of group10(G98, G99)
0	9	1	When the power off, G mode of group14(G54～G59)
0	9	2	The value of F when the power off.
0	9	3	The value of S when the power off.
0	9	4	The value of H when the power off.
0	9	5	The value of D when the power off.
1	0	6	Allowed Max. spindle speed when rigid tapping
1	0	7	Counts of X pulse from checking PC to receving PC in Ref.
1	0	8	Counts of Y pulse from checking PC to receving PC in Ref.
1	0	9	Counts of Z pulse from checking PC to receving PC in Ref.
1	1	0	Counts of 4th pulse from checking PC to receving PC in Ref.
1	1	1	Counts of 5th pulse from checking PC to receving PC in Ref.

1	1	2	The pulse counts of spindle encoder

1	1	3	The pulse counts of handwheel

Note:：In fixed cycle program,№079～№082 means the current section's start position,but not the program segment's start position,when power off.

5.2 PLC state

This part of diagnosis is used to detect the signal state of machine→PLC（X）, PLC→machine（Y）,CNC→PLC（F）,PLC→CNC（G）and alarm address A, which can't be modified. See the relative PLC manual for address F, G significance, and the signal significance of address A is defined by user himself.

5.2.1 X address (fixed addresses)

X0000			ESP		DECX			

ESP：Emergency stop signal

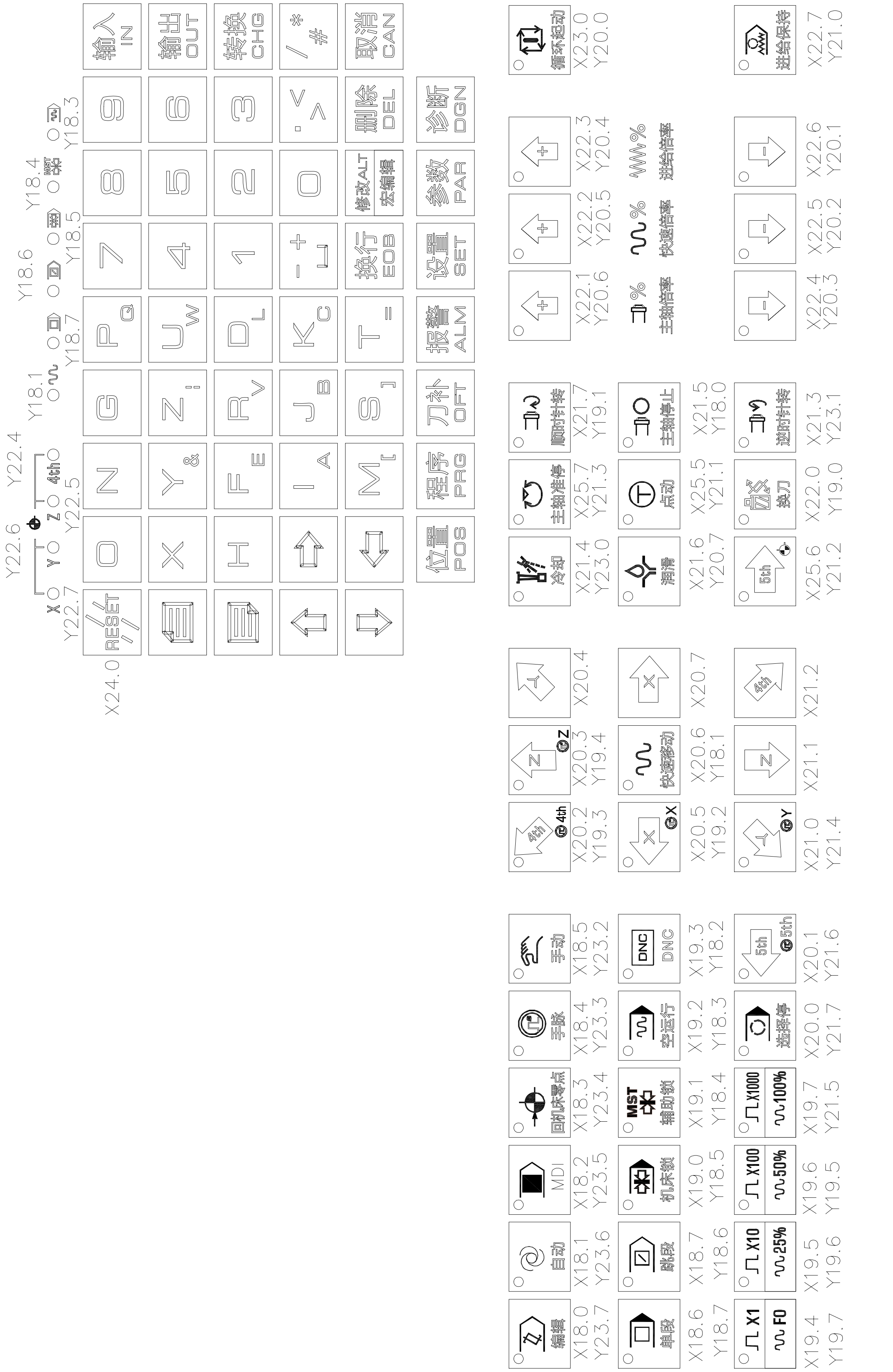
DECX：Deceleration signal of X axis

X0001					DECZ			
X0002			DEC5	DEC4	DECY			
X0003			SKIP					

SKIP：Skip signal

DECY~DEC5：Deceleration signal of (Y, Z, 4th, 5th) axis

Corresponding machine panel keys to X fixed address, refer to the following figure:



Volume Ⅲ Installation

5.2.2 Y address (fixed addresses)

Corresponding machine panel and state indicator to Y fixed address, refer to the above figure:

5.3 PLC Data

The PLC data includes T, C, DT, DC, D, their significance is defined by user requirement.

CHAPTER 6 MEMORIZING SCREW-PITCH ERROR COMPENSATION FUNCTION

6.1 Function Explanation

There are more or less precision errors in the screw-pitch of machine axes lead screw, it will definitely affect the parts machining precision. This GSK980MD has the memorizing screw-pitch error compensation function that it can accurately compensate the screw-pitch error.

6.2 Specifications

1 The offset is concerned with the offset origin, offset clearances, offset point, mechanical moving direction etc.;

2 After performing the machine zero return, take this reference point as the offset origin, and set the offset value into the parameters according to axes compensation intervals;

3 Points to be compensated: 256 points for each axis

4 Axis to be compensated：X, Y, Z，4th, 5th axis

5 Offset range: -255～+255μ m for each offset point

6 Offset clearance: 1000～9999999μm;

7 Offset of point N (N=0, 1, 2, 3,…255) is determined by the N, N-1 mechanical error;

8 Actual offset interval: set an appropriate value in the range above according to the max. offset range and mechanical travel;

9 The setting is the same as the CNC parameters input, see the explanation in the relative operation.

6.3 Parameter Setting

6.3.1 Screw-pitch compensation

0	0	3		***	***	PCOMP	***	***	***	D/R	***

PCOMP =1: Screw-pitch error compensation valid;

=0: Screw-pitch error compensation invalid.

6.3.2 Screw-pitch error origin

A position No. which the screw-pitch error compensation starts from in the compensation list, which is determined from the machine zero, is called screw-pitch error compensation origin (compensation original point). Each axis may be set in any position from 0 to 255, which is set by data parameter №125～№129 depending on the mechanical requirement.

1	2	5	Screw-pitch error offset No. of X machine zero
1	2	6	Screw-pitch error offset No. of Y machine zero
1	2	7	Screw-pitch error offset No. of Z machine zero
1	2	8	Screw-pitch error offset No. of 4th machine zero
1	2	9	Screw-pitch error offset No. of 5th machine zero

6.3.3 Offset interval

1	2	0	Clearance of X axis screw-pitch offset
1	2	1	Clearance of Y axis screw-pitch offset
1	2	2	Clearance of Z axis screw-pitch offset
1	2	3	Clearance of 4th axis screw-pitch offset
1	2	4	Clearance of 5th axis screw-pitch offset

Setting range： 1000～999999（unit：0.001mm ）

6.3.4 Compensation value

The axes screw-pitch offset values are set in the page of screw-pitch parameter. Refer to the following table.The offset value is input by diameter with the unit 0.001mm, which is irrelevant to the programming by diameter or by radius. （Take X, Y, Z axes as example）

Offset No.	**X**	**Y**	**Z**
000	…	…	…
001	5	-2	3
002	-3	4	-1
…	…	…	…
255	…	…	…

6.4 Cautions for Offset Setting

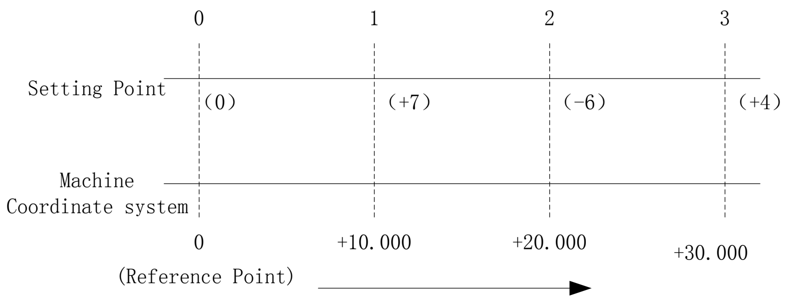
① The setting and modification of screw-pitch offset can only be done at the authority of password level 2 and switch on parameter switch.
② Offset is not allowed if the offset interval entered is 0
③ After the parameter of screw-pitch offset is set, only the machine zero is returned could the compensation be done.

6.5 Examples of Offset Parameters Setting

① parameter №125（screw-pitch error origin point）=0, Data parameter №120（screw-pitch offset interval）=0,

When the screw-pitch error origin is set to 0: The offset value for the 1st section is set in screw-pitch compensation parameter list №001, the offset value for the 2nd section is set in screw-pitch compensation parameter list №002, and the offset value for the Nth section is set in screw-pitch compensation parameter list №(000+N).

The machine zero is regarded as the reference point of screw-pitch error origin point; it begins to compensate the position №001 in the offset table from the machine zero. So the screw-pitch error compensation can only be performed in the positive moving of the machine zero coordinate system.



The position No.000 in the offset table corresponds to the reference point (i.e screw-pitch error origin 0), the offset point 1 corresponds to a point 10.000 positive moving from this reference point, and there is a compensation point from this point every 10.000 distance. The 127th compensation point is the offset value at position 1270.000. Therefore, at compensation point 1, set an compensation value moving from 0 to 10.000, at offset point 2, set an offset value moving from 10.000 to 20.000. At offset point N, set an offset value moving from (N-1) × (offset clearance) to N × (offset clearance).

Above is the example of following offset interval errors：

Offset clearance	**Offset value**
0～10.000	+7
10.000～20.000	-6
20.000～30.000	+4

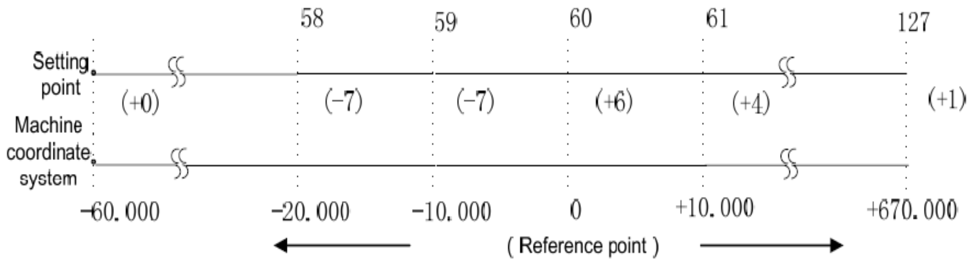
Machine coordinate system	**Offset parameter No.**	**Offset value**	**Drive unit current command pulses before offsetting**	**Drive unit current command pulses after offsetting**
Reference point 0	000	000	00000	00000
10.000	001	7	10000	10007
20.000	002	-6	20000	20001
30.000	003	4	30000	30005
……	004	…		

② Data parameter№125（screw-pitch error origin）=60，№0120（compensation interval）=10.000

When the screw-pitch error origin is set to 60: For the positive moving, the compensation value for the 1st section is set by the position №061 in the compensation table. The compensation value for the 2nd section is set by the position №062 in the compensation table. The compensation value for the Nth is set by position №060+N in the compensation table.

For the negative moving, the 1st section error compensation is set by position №060 in the compensation table, the 2nd section by position №059. The Nth section error compensation is set by position №060-N in the compensation table.

By taking the machine zero as the reference point, the screw-pitch error origin moves from the positive coordinate system of machine zero to compensate the corresponding position No.061 in the compensation table, and from the negative coordinate system to compensate the position No.060. Therefore the screw-pitch compensation can be done when moving in the positive or the negative coordinate system of machine zero.



The position No.060 in the screw-pitch error compensation parameters corresponds to the reference point (60), compensation point 61 to a point positive10.000 moving from origin. So there is a compensation point every 10.000 distance. The 127th offset point is the compensation at position +670.000. While thecompensation point 59 corresponds to a point negative 10.000 moving from reference point. Also there is a compensation point every 10.000 .The offset point 0 is the compensation value at -600.000 position. Therefore, at compensation point N, set a compensation valuewhen moving from (N-61) × (compensation interval) to (N-60) × (compensation interval).

Above is the example of following compensation interval errors

Offset interval	Offset value
0～10.000	+4
-10.000～0	+6
-20.000～-10.000	-7
-30.000～-20.000	-7

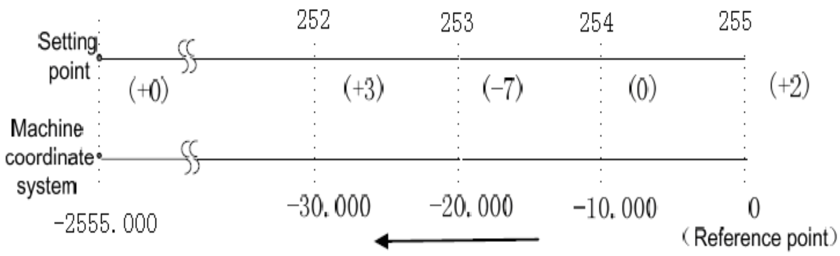
Machine coordinate system	Offset parameter No.	Offset value	Drive unit current command pulses before offsetting	Drive unit current command pulses after offsetting
-30.000	058	-7	-30000	-29992
-20.000	059	-7	-20000	-19999
-10.000	060	+6	-10000	-10006
Reference point 0			0	0
10.000	061	+4	10000	10004
……	062	…		

Actually the machine moves from -30.000 point to the point of +10.000, the screw-pitch compensation is: (-7)+(-7)+(+6)+(+4)=(-4)

③ Data parameter №125（screw-pitch error origin）=255，№120（compensation interval）=10000

When the screw-pitch error origin is set to 255: The compensation value for the 1st section is set by the position №255 in the compensation table, the compensation value for the 2nd section is set by the position №254 in the compensation table, and the compensation value for the Nth section is set by the position №256-N in the compensation table.

The machine zero is regarded as the reference point of screw-pitch error origin. It begins to compensate the position №255 in the compensation table from the machine zero. So the screw-pitch error compensation can only be done in the negative moving of the machine zero coordinate system.



The compensation point 254 corresponds to a point moving 10.000 in negative direction from the reference point.There is a compensation point every -10.000 distance. Compensation point 1 is the compensation value at position -1260.000. Therefore, set an offset value moving from 0 to -10.000 at compensation point 255; set an offset value moving from -10.000 to -20.000 at offset point 254. At compensation point N, set an offset value moving from (N-256)×(compensation iterval) to (N-255)×(compensation interval).

The above is the example of following compensation interval errors:

Compensation interval	Compensation value
0～-10.000	+2
-20.000～-10.000	0
-30.000～-20.000	-7
-40.000～-30.000	+3

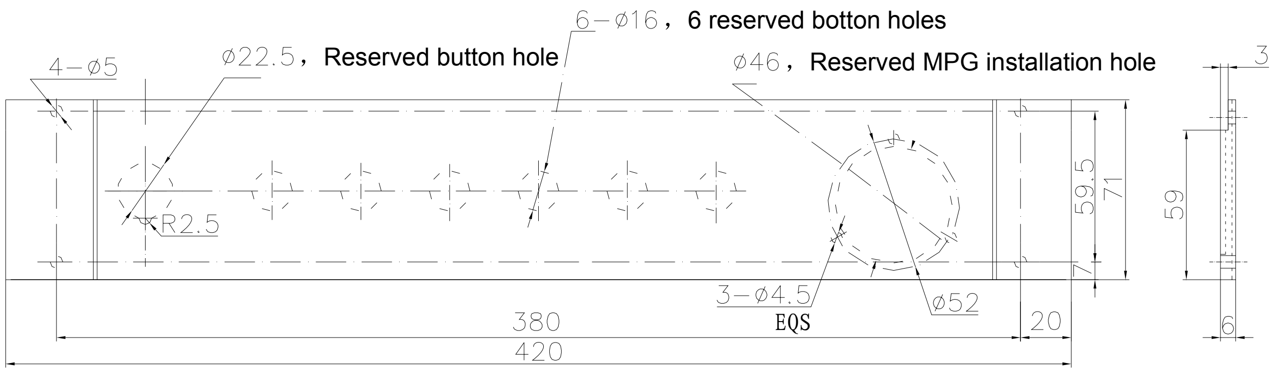
Machine coordinate system	Compensation parameter No.	Compensation value	Drive unit current command pulses before offsetting	Drive unit current command pulses after offsetting
Reference point 0			0	0
-10.000	255	2	10000	10002
-20.000	254	0	20000	20002
-30.000	253	-7	30000	29995
-40.000	252	3	40000	39998

Actually the machine moves from the point -40.000 to the reference point, the screw-pitch compensation is: (+3)+(-7)+(0)+(+2)=(-2)

Appendix

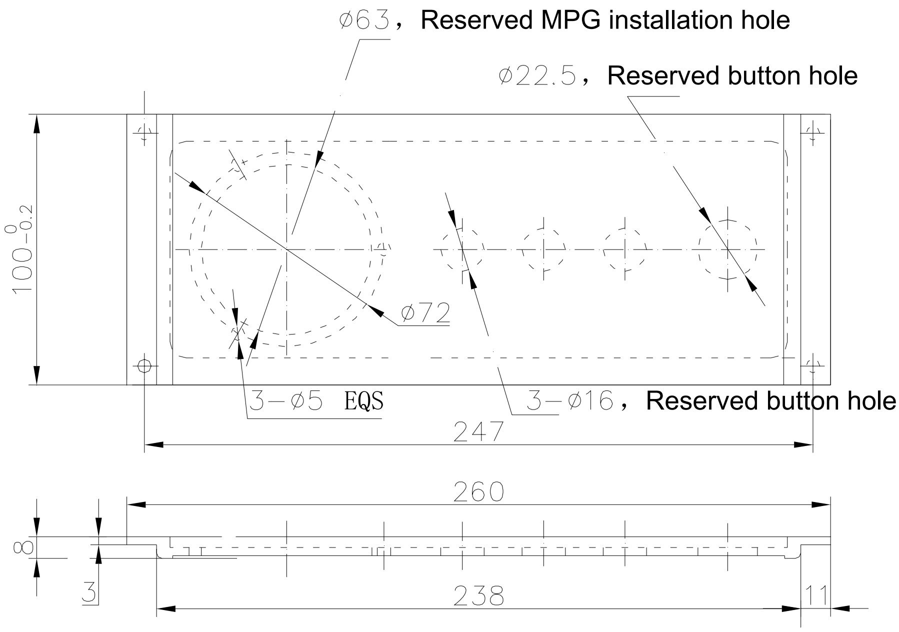
Appendix 1.   Dimensions of Additional Panel AP01

AP01: Aluminum alloy 420mm×71 mm, it can be mounted below the panel. Its figure and dimensions are as follows:



Appendix 2   Dimensions for Additional Panel AP02

AP02: Aluminum alloy 100mm×260mm, it can be mounted to the side of the panel, its figure and dimensions are as follows:



APPENDIX

Appendix 3 GSKComm Introduction (leading-in from winapp)

Appendix 4 Alarm Information

№.	CONTENTS
000	Emergency stop, ESP open circuit
001	The called program does not exist or is failed to open
002	G code is illegal
003	Total characters of single command is out of range (2~12 characters are allowed)
004	Specified pitch F value exceeds permitted range
006	The specified block number format is wrong (not at the beginning or repeated),or the number exceeds permitted range
008	Addresses I,J,K for the center of a arc in arc command is specified to axes that beyond the selected plane.
009	G command of group 00 or 01 is not input and a movement value is specified when there is no valid G command of 01 group.
010	One command address is repeated in the same block
011	The input words in the same block are more than 100
012	The command value is illegal or out of permitted range
013	S command out of range S00~99 is input when spindle analog voltage control is inactive
014	when G codes of group 00 and group 12 are commanded, G codes of group 01 can not be specified in the same block.
015	M command for automatic gear change are commanded when spindle analog voltage control is inactive
016	The specified sequence number of tool length compensation is wrong or out of range(0~32)
017	Tool number is out of range set by parameter NO. 213
018	Arc data is wrong (or exceeds the permitted radius error set by parameter NO. 175)
019	Tool group number excesses the range (1~32) in tool life management function
020	Tool radius compensation number exceed the range(0~32)
021	The value of I 、J or K is not correct in G02 or G03 command
022	Additional axes (4th,5th axis) can not perform circular interpolation
023	The specified value at F address is wrong or exceeds the range set by parameter NO.070
024	G11 is absent in the program
025	No tool is found in the current tool group in tool life management function
026	The current tool group is undefined in tool life management function
027	The number of tool in the current tool group exceeds 8 in tool life management function
028	tool life management function is invalid, command G10L3 is unallowable.
029	G11 can not be programmed before G10
030	The plane is changed when tool radius compensation is performed
031	Plane switch, tool length compensation and coordinate change cannot be performed

APPENDIX

	when chamfering
032	R value is specified in circular command, but the amount of movement in the arc plane is not specified.
033	When a radius is set or changed, the corresponding movement amount must be specified.
034	The arc data is wrong in radius compensation (or compensation mode is wrong).
035	G31 cannot be used in radius compensation
036	Format of chamfer is wrong (or the subsequent block is not G01,G02,G03)
037	The number of character is more than 256 in one block.
038	The format of G20 and G21 is wrong (The switch of inch or metric system must be headed.)
039	Radius compensation cannot be set together with chamfering.
040	G40～G42,G140～G143 are disabled in single block mode(MDI)
041	The format of annotation is wrong (unclosed bracket)
042	G02,G03,G04,G31,G92,G142,G143 cannot be in the same block with G43,G44,G49,H
043	Result in a macro program is out of range (data overflow)
050	Skip is disabled in DNC mode
095	No sequence number is input or illegal sequence number is commanded in subprogram call
096	The depth of nested subprogram exceeds 4 levels
097	The called program is the current program (main program)
098	Macro call or M98 and M99 cannot be commanded in single block mode (MDI)
099	Macro can not be called ,or M98 and M99 can not be commanded during radius compensation
100	Skip(GOTO,DO,END) is disabled during radius compensation
101	The format of macro statement is wrong.
102	The label of DO or END is not 1、2 or 3 in a macro statement
103	The format of DO or END is wrong in a macro statement (or jump into a cycle)
104	Bracket unclosed or format error occurs in a macro statement
105	The divisor should not be zero in a macro statement.
106	The format of ATAN is wrong
107	The inverse logarithm of LN is wrong (≤0).
108	The radicand value should not be negative.
109	The result of TAN is a infinitude
110	The operand of ASIN or ACOS is out of range (<-1,or >1).
111	The variable type is wrong or inexistent.
112	The block number called by GOTO or M99 is overflow or not exist.
113	M98 or M99 can not be executed when G66 is commanded.
114	G65 or G66 must be defined ahead of a block.
115	G65 cannot in the same block with G43,G44,G49
116	G65 cannot in the same block with M00,M01,M02,M30,M98,M99
117	Null variable cannot be the result in a macro statement
118	The number of argument I, J,K in G65 and G66 exceeds 10
119	Macro program called by P in G65,G66 is out of permitted range

120	The read-only variables in macro statement cannot be written
121	The value assigned to system variable cannot be null
122	P value is not specified to G65 or G66, or H calculation is not specified by G65
123	G65 H_ format is wrong
124	Illegal H command is specified in G65
125	Proper operands and number are not specified in macro
126	Alarm number specified by G65 H99 is beyond range (P:0～99)
127	Operand is not integer to convert into binary form in macro statement
128	Operand is not binary number in macro statement
129	Radius compensation is in pre-read mode, thus correct macro variable is not available.
131	validate the rotary axis before using CS axis
132	Operand value is too large in macro statement
205	K value is undefined
206	I value is undefined
207	I value is too small
208	J value is undefined
209	J value is too small
210	U value is too large, or I,J is too small
211	J value is too large
212	K value is too small
213	U value is less than tool radius
214	I, J is too small or K is too large, which result in overcut.
215	During continuous rectangular drilling, J value is not specified or the end point and start point of the first side is identical, which makes it impossible to identify a rectangle
216	no drill mode (G73～G89) is specified for G140~G143 continuous drilling
217	drill holes cannot be less than 2
218	Pitch F value is not specified in G74, G84
219	drill interval is too small in canned cycle
230	S value is 0 and spindle feed is disabled.
231	S value is beyond the maximum rotation speed for rigid tapping
232	other movement along axes is specified between M29 and G74/G84
233	G61.0 rigid tapping signal is abnormal
234	M29 repeated
235	When positioning is needed for command M29, it should not be in the same block with G74,G84
236	The 5th axis should be specified as rotation axis before rigid tapping
250	Radius compensation cannot be performed due to the coincidence of compensation start point and circular start point
251	Programming error result in mistake in radius compensation calculation
252	Programming error causes the end point of a machined arc is not on the arc
253	Programming error. The coordinates of two adjacent points are coincident, which leads to invalid radius compensation
254	Programming error. Superposition of the centre and start point of the arc leads to invalid radius compensation

255	Programming error. Superposition of the centre and end point of the arc leads to invalid radius compensation
256	As arc radius is smaller than tool nose radius, radius compensation cannot be performed
257	Programming error. No intersection between two arcs with current tool compensation C
258	G02,G03 is commanded when establishing tool compensation C
259	G02,G03 is commanded when canceling the tool compensation C with G40
260	Overcut is found in the interference checking for the tool compensation C
261	Programming error result in no intersection between the line and arc path in tool compensation C
262	Programming error result in no intersection between the arc path and line in tool compensation C
263	Overflow of cutter compensation buffer due to excessive non-movement commands
281	Length of linear chamfer is excessive
282	Radius of arc chamfer is excessive
283	Length of linear chamfer is excessive or arc data is wrong
284	Radius of arc chamfer is excessive or arc data is wrong
287	Length of linear chamfer is excessive or the intersection point is not no the arc
288	Radius of arc chamfer is excessive or the intersection point is not on the arc
289	The start point and end point are coincident in the arc chamfer plane, which disable the chamfering
301	Parameter switch is ON
302	CNC initialization failure
303	Part program open failure
304	Part program saving failure
305	New part program creating failure
306	Illegal word is input
307	Insufficient memory capacity
308	Program number is out of range
309	Macro program writing is prohibited by current operation authority
310	PLC program open failure
311	Software version is not conformed to PLC program
312	The first grade program for PLC program is too long
313	Fault occurs in keyboard or panel
314	Fault occurs in storage, check it and repower-on
315	Fault occurs in DNC com, check the hardware connection and baudrate
316	parameter file saving failure
317	System file error
318	Text format error
319	file pointer error in program loading
320	File pointer positioning error during program loading
321	File read error during program loading
322	Program location error
323	Invalidate the rotation axis before using Cs contouring control
324	names of 4th and 5th axis cannot be the same

325	2 Cs axes cannot be validate at the same time. Modify the parameter.
326	CNC file deletion failure
327	USB files reading and writing error (connect it again)
328	Files copy error
329	Files re-download error
330	Program loading failure (the length of block is excessive—more than 255 characters)
350	Parameter files open failure. Set standard parameter at the factory
351	Parameter loading error. Set it as standard at the factory
352	Data check error occurs in data retention area. Restore the area and operate after zero return.
353	Data check error occurs in data retention area. Standard value is restored. Operate after zero return.
354	Alteration of parameter is valid after re-power-on
355	System upgrade and renew is finished, re-power-on to validate it
356	Current ladder diagram is changed, please re-power-on
357	CNC files are in use, confirm the machining programs
359	Alteration of serial communication parameter is valid after re-power-on
360	The least increment system is changed, please re-power-on and check the range and setting of corresponding parameters.
361	The least increment for additional axes should not be less than the least increment of the system (IS-B,IS-C)
362	The velocity parameter is out of permitted range. Modify it.
363	The exceeded velocity parameter is modified. Operate the machine tool after confirmation
401	The intermediate point is not specified in G29
402	The highest rotation speed is not defined, please check the parameter No.101~No.104
403	Feedrate is too high
404	Feeding stop because spindle stop.
405	Spindle rotation speed is too slow for thread cutting
406	Spindle rotation direction is not same with specified direction
407	Spindle speed fluctuation exceeds the range set by NO.108
408	Spindle mode switching is prohibited when Cs axis is moving
409	Reference point is not set, return to 2nd or 3rd or 4th reference point is disabled
410	CS axis movement disabled as spindle is not in position control mode
411	Exceeds the X axis + side software stroke limit
412	Exceeds the Y axis + side software stroke limit
413	Exceeds the Z axis + side software stroke limit
414	Exceeds the 4th axis + side software stroke limit
415	Exceeds the 5th axis + side software stroke limit
416	Exceeds the X axis - side software stroke limit
417	Exceeds the Y axis - side software stroke limit
418	Exceeds the Z axis - side software stroke limit
419	Exceeds the 4th axis - side software stroke limit
420	Exceeds the 5th axis - side software stroke limit

421	Overtravel along X axis positive direction
422	Overtravel along Y axis positive direction
423	Overtravel along Z axis positive direction
424	Overtravel along 4th axis positive direction
425	Overtravel along 5th axis positive direction
426	Overtravel along X axis negative direction
427	Overtravel along Y axis negative direction
428	Overtravel along Z axis negative direction
429	Overtravel along 4th axis negative direction
430	Overtravel along 5th axis negative direction
431	X axis driver is not ready
432	Y axis driver is not ready
433	Z axis driver is not ready
434	4th axis driver is not ready
435	5th axis driver is not ready

Appendix 5 Function Configuration of Standard Ladder Diagram

5.1 Information for Ladder Diagram

5.1.1 Introduction

Rang of use: Configuration for standard ladder diagram

Software version: Standard

5.1.2 Information of Current Version

DESIGN:GSK

VERSION: 09.08.13-91F5

VERIFY: 91F5

REMARK: GSK980MDa Standard Ladder Diagram

5.2 ADDRESS DEFINITION

CN61	PLC address	Standard PLC address definition	Function for standard PLC address definition	Remark
1	X0.0			
2	X0.1	SP	External dwell signal	
3	X0.2			
4	X0.3	DECX	X axis deceleration signal	Fixed address
5	X0.4			
6	X0.5	ESP	External emergent stop signal	Fixed address
7	X0.6			
8	X0.7			
9	X1.0			
10	X1.1			
11	X1.2			
12	X1.3	DECZ	Z axis deceleration signal	Fixed address
13	X1.4	ST	External cycle start signal	
14	X1.5			
15	X1.6	SPAL	Spindle alarm signal	
16	X1.7			
29	X2.0			
30	X2.1			
31	X2.2			
32	X2.3	DECY	Y axis deceleration signal	Fixed address
33	X2.4	DEC4	4th axis deceleration signal	Fixed address
34	X2.5	DEC5	5th axis deceleration signal	Fixed address
35	X2.6			
36	X2.7			
37	X3.0			
38	X3.1			
39	X3.2			
40	X3.3			
41	X3.4			
42	X3.5	SKIP	Skip signal	Fixed address
43	X3.6			
44	X3.7			
17~20	Null			
21~24	GND			
25~28	Null			

CN62	PLC address	Standard PLC address definition	Function for standard PLC address definition	Remark
1	Y0.0	COOL	Cooling signal	
2	Y0.1	LUBR	Lubricating output signal	
3	Y0.2			
4	Y0.3	SFR	Spindle CCW signal	
5	Y0.4	SRV	Spindle CW signal	
6	Y0.5	SSTP	Spindle stop signal	
7	Y0.6	ENB	Spindle enable signal	
8	Y0.7	SPZD	Spindle braking signal	
9	Y1.0	GEAR1	Spindle mechanical gear signal 1	
10	Y1.1	GEAR2	Spindle mechanical gear signal 2	
11	Y1.2	GEAR3	Spindle mechanical gear signal 3	
12	Y1.3	GEAR4	Spindle mechanical gear signal 4	
13	Y1.4			
14	Y1.5			
15	Y1.6			
16	Y1.7			
29	Y2.0			
30	Y2.1			
31	Y2.2	CLPY	Yellow Lamp	
32	Y2.3	CLPG	Green Lamp	
33	Y2.4	CLPR	Red Lamp	
34	Y2.5			
35	Y2.6			
36	Y2.7	ALTO	ALT.output signal	
37	Y3.0			
38	Y3.1			
39	Y3.2			
40	Y3.3			
41	Y3.4			
42	Y3.5			
43	Y3.6			
44	Y3.7			
17~19	GND			
20~25	+24V			
26~28	GND			

CN31	PLC address	Standard PLC address definition	Function for standard PLC address definition	Remark
5	X6.0	EHDX	External MPG X axis choosed	
6	X6.1	EHDY	External MPG Y axis choosed	
8	X6.2	EHDZ	External MPG Z axis choosed	
9	X6.3	EMP0	External X1 override	
22	X6.4	EMP1	External X10 override	
23	X6.5	EMP2	External X100 override	
1	HA+			
2	HA-			
3	HB+			
4	HB-			
7、19~21	NULL			
24~26				
10~13	GND			
14~16	+5V			
17~18	+24V			

CN15	PLC address	Standard PLC address definition	Function for standard PLC address definition	Remark
5	X5.0			
6	X5.1	VPO	spindle V/P output signal	
8	X5.2			
20	Y5.0	VP	Spindle V/P switch signal	
21	Y5.1	TAP	rigid tapping signal	
22	Y5.2			
23	Y5.3			
3、9、12、16、25	GND			
11、17	+24V			

Refer to Book 3—chapter 5 Diagnosis Information for panel key-press and panel indicator lights addresses.

5.3 FUNCTION CONFIGURATION

5.3.1 Spindle CCW and CW Control

- **Relevant signals**

Type	Sign	Meaning	Corresponding pin-out	PLC state	CNC diagnosis
Input signal		The CW key on the machine panel		X21.7	
		The CCW key on the machine panel		X21.3	
		The Stop key on the machine panel		X21.5	
	SPAL	Spindle alarm signal	CN61.15	X1.6	
Output signal	ENB	Spindle enable signal	CN62.7	Y0.6	
	SFR	CCW signal	CN62.4	Y0.3	
	SRV	CW signal	CN62.5	Y0.4	
	SSTP	Spindle stop signal	CN62.6	Y0.5	
	SPZD	Spindle braking signal	CN62.8	Y0.7	
		CCW indicator on the machine panel		Y23.1	
		CW indicator on the machine panel		Y19.1	
		Spindle stop indicator on the machine panel		Y18.0	
Command input	M03	Command signal for CCW			
	M04	Command signal for CW			
	M05	Command signal for spindle stop			

- Control Parameter

K0010								RSJG	

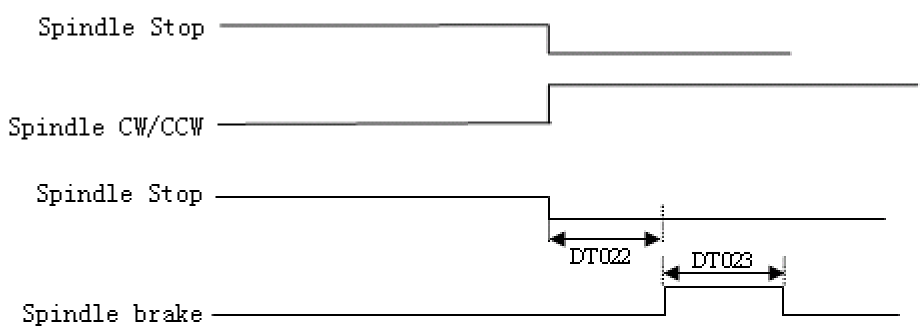
RSJG =1: Output signals for spindle, cooling and lubricating are on when resetting

=0: Output signals for spindle, cooling and lubricating are off when resetting

DT0021	**M code execution time**
DT0022	Delay time of spindle stop to braking output
DT0023	Spindle braking output time

- **Operation Sequence**

Spindle operation sequence is as follows:



Note: DT022 is the delay time from the spindle stop signal issued to spindle brake signal is issued; DT023 is spindle brake hold time.

- **Logic control**

SSTP output is validated after the CNC is power-on. When SSTP output is valid, the executing of M03 or 04 will disable SSTP output and enable SFR or SRV output. The executing of M05 will disable SFR or SRV output and enable SSTP output. Spindle braking signal SPZD output delay is set by data DT022 of PLC (The delay time between the spindle stop command output and braking signal SPZD output). The time for braking signal hold-on is set by data DT023 of PLC (Spindle brake output time).

If the spindle rotates in CCW or CW, the alarm A0.3: "Illegal command M03 or M04" is generated if the M04 or M03 is commanded.

Note 1: When the CNC external emergency stop or spindle alarm is issued, the spindle rotation output signal is off, and meanwhile the SSTP signal is output.

Note 2: When CNC is reset, whether SRV or SFR output is cancelled is determined upon the bit (RSJG) of K0010:

When RSJG = 0, SRV or SFR output is closed after the CNC is reset.

When RSJG = 1, SRV or SFR output state unchanged after the CNC is reset.

Note 3: The alarm A0.1 (spindle alarm) is generated when the spindle alarm signal X1.6 is detected by PLC;

Note 4: In the spindle analog volume control, if the output voltage is more than 0; the spindle enable signal is valid.

5.3.2 Spindle JOG

- **Relevant signals**

Type	Sign	Meaning	Corresponding pin-out	PLC state	CNC diagnosis
Input signal		Signal for spindle JOG key on machine panel		X25.5	
Output signal		Indicator for spindle JOG start-up on machine panel		Y21.1	

- **Control parameter**

K0010					JSPD				

JSPD =0: Spindle JOG is effective only in JOG、MPG、REF modes

=1: Spindle JOG is effective in any mode.

- **Function description**

When holding down the Spindle JOG key on the machine panel, the spindle rotates CCW and it stops rotating as soon as the key is released.

5.3.3 Switch Value Control for Spindle Speed

- **Relevant signals**

Type	Sign	Meaning	Corresponding pin-out	PLC state	CNC diagnosis
Output signal	GEAR1	Spindle mechanical gear signal 1	CN62.9	Y1.0	
	GEAR 2	Spindle mechanical gear signal 2	CN62.10	Y1.1	
	GEAR 3	Spindle mechanical gear signal 3	CN62.11	Y1.2	
	GEAR 4	Spindle mechanical gear signal 4	CN62.12	Y1.3	
Command input	S01	Command signal for spindle gear signal 1			
	S02	Command signal for spindle gear signal 2			
	S03	Command signal for spindle gear signal 3			
	S04	Command signal for spindle gear signal 4			
	S00	Command signal for spindle gear signal cancellation			

- Control parameter

0	0	1					ACS				
Corresponding F signal							F200.4				

ACS =1: Analog voltage control for spindle speed;

=0: Switch value control for spindle speed.

0	1	8					ESCD				
Corresponding F signal							F211.4				

ESCD =0: The S code not closed when stopping urgently;

=1: The S code closed when stopping urgently.

DT0019	S code performance time
DT0024	Delay time for spindle gear shift

APPENDIX

- Logic control

  The GEAR1~GEAR4 output are disabled when CNC is power-on. When any of commands S01, S02, S03 and S04 being executed, the corresponding S signal output is validated, and the rest 3 S signals output is cancelled at the same time. When the S00 command is executed, the GEAR1~GEAR4 output are cancelled, and only one of them is effective at the same time.

5.3.4 Cycle Start and Feed Hold

- **Relevant signals**

Type	**Sign**	**Meaning**	**Pin-out**	**PLC state**	**CNC diagnosis**
Input signal	ST	External cycle start signal	CN61.13	X1.4	
	SP	External dwell signal	CN61.2	X0.1	
		Cycle Start key signal on the machine panel		X23.0	
		Feed Hold key signal on the machine panel		X22.7	
		OUT cycle start signal on MDI panel		F197.1	
Output signal		Cycle start indicator on machine panel		Y20.0	
		Feed hold indicator on machine panel		Y21.0	
Command input	M00	Feed hold command		F9.7	

- **Control Parameter**

0	1	7		MST	MSP					
Corresponding F signal				F210.6	F210.5					

MST =1: External cycle start signal disabled;

=0: External cycle start signal enabled;

MSP =1: External feed hold signal disabled;

=0: External feed hold signal enabled, and external dwell switch needed, or CNC “dwell” alarm occurs;

K0010							OUTR		

OUTR =1: Program runs by OUT key on the MDI panel in MDI mode.

=0: Program runs by OUT key on the MDI panel disabled in MDI mode

- Note

  Normal opened bottom without auto-lock is needed for external cycle signal ST; Normal closed bottom without auto-lock is needed for external dwell signal SP;

5.3.5 Cooling Control

- **Relevant signals**

Type	Sign	Meaning	Pin-out	PLC state	CNC diagnosis
Input signal		Cooling key signal on machine panel		X21.4	
Output signal		Cooling starts indicator on machine panel		Y23.0	
	COOL	Cooling output signal	CN62.1	Y0.0	
Command input	M08	Command signal for cooling starts			
	M09	Command signal for cooling off			

- **Control parameter**

K0010								RSJG	

RSJG =1: M03, M04, M08 and M32 output signals are turned off when resetting.

=0: M03, M04, M08 and M32 output signals are turned on when resetting.

- **Function description**

COOL is disabled after CNC is power-on, COOL output is effective and the cooling pump is ON when M08 is executed; COOL output is cancelled if M09 is executed, and the cooling pump is OFF.

Note 1: ON/OFF state of cooling output is defined by the RSJG of K10 when CNC is reset;

Note 2: M09 has no corresponding output signal; the output of M08 is cancelled as M09 is executed.

Note 3: The cooling output is OFF when M30 is executed.

5.3.6 Lubricating control

- **Relevant signal**

Type	Sign	Meaning	Pin-out	PLC state	CNC diagnosis
Input signal		Lubricating key signal on the machine panel		X21.6	
Output signal		Lubricating on indicator on machine panel		Y20.7	
	LUBR	Lubricating output signal	CN62.2	Y0.1	
Command input	M32	Lubricating on command signal			
	M33	Lubricating off command signal			

- **Control parameters**

DT0016	time of auto lubrication cancel
DT0017	0：not auto lubrication,>0：auto lubrication
DT0018	not auto lubr..0：alternative lubrication,>0：scheduler lubrication

APPENDIX

- **Function description**

There are two lubricating functions defined by GSK980MD standard PLC program: non-automatic lubricating and automatic lubricating. They are set by PLC data.

DT017 =0: For non-automatic lubricating

>0: For automatic lubricating, lubricating time DT017 and lubricating interval time DT016 can be set.

DT018 =0: For non-automatic lubricating, lubricating reverses.

>1: For non-automatic, timing lubricating.

1. Non-automatic lubricating function

When PLC data DT018 is equal to 0, it is lubricating reverse output. Lubricating output is performed by pressing the Lubricating key on the machine panel; lubricating output is cancelled if the key is pressed again. When M32 is performed and the lubricating is output; cancel the lubricating output by executing M33.

When the PLC data DT018>1, it is lubricating timing output. Lubricating output is performed when the Lubricating key on the machine panel is pressed. The lubricating output is cancelled when the period set by DT018 elapsed. So, M32 is performed for the lubrication output, it is cancelled after the period set by DT018 elapsed. If the period set by DT018 has not elapsed, M33 is performed then the lubricating output is cancelled

2. Automatic lubricating

Lubricating starts after power-on, as the period set by DT017 elapsed, the output stops. As the period set by DT016 elapsed, lubricating is valid again. This process is repeated. The M32, M33 commands and the Lubricating key on the machine panel are all ineffective when the automatic lubricating is applied.

5.3.7 Optional Block Skip

The optional block skip function can be applied when one block is neither performed nor deleted in a program. When the "/" is at the head of a block and the optional block skip switch is ON (the Block Skip key on machine panel or external output for optional block skip is enabled), this block is not run in automatic operation

- Relevant Signals

Type	Sign	Meaning	Pin-out	PLC state	CNC diagnosis
Input signal		optional block skip key signal on machine panel		X18.7	
Output signal		Optional block skip indicator on machine panel		Y18.6	

- Function description

1. When optional block skip signal is effective, a block headed with "/" is not performed.
2. The optional block skip function is enabled only in Auto, MDI and DNC modes

APPENDIX

5.3.8 Machine Lock

- Relevant signals

Type	Sign	Meaning	Pin-out	PLC state	CNC diagnosis
Input signal		Machine lock key signal on the machine panel		X19.0	
Output signal		Machine lock indicator on the machine panel		Y18.5	

- Function description

1. Machine locks are enabled in any mode.
2. The machine lock state can not be shifted when program is running.

5.3.9 MST Lock

Relevant signals

Type	Sign	Meaning	Pin-out	PLC state	CNC diagnosis
Input signal		MST lock key signal on the machine panel		X19.1	
Output signal		MST lock indicator on machine panel		Y18.4	

- Function description

MST lock is enabled in Auto, MDI or DNC modes；

5.3.10 Single Block

- Relevant signal

Type	Sign	Meaning	Pin-out	PLC state	CNC diagnosis
Input signal		Single key signal on machine panel		X18.6	
Output signal		Single indicator on machine panel		Y18.7	

- Function description
- Single block is enabled in Auto, MDI or DNC mode；

5.3.11 Dry Run

- Relevant signals

Type	Sign	Meaning	Pin-out	PLC state	CNC diagnosis
Input signal		Dry run key signal on machine panel		X19.2	
Output signal		Dry run indicator on machine panel		Y18.3	

- Function description

  1. Dry run for program is enabled in Auto, MDI or DNC modes.
  2. Dry run state can not be switched during the program execution.

5.3.12 Optional Stop

- Relevant signal

Type	Sign	Meaning	Pin-out	PLC state	CNC diagnosis
Input signal		optional stop signal on machine panel		X20.0	
Command input	M01	optional stop command		F9.6	
Output signal		optional stop indicator on machine panel		Y21.7	

- Function description

In Auto, MDI and DNC modes, press key 

to light up the optional indicator, which means the system enters into optional stop state. When command M01 is executed, “dwell” will be performed. The program will continue by pressing key 

again.

5.3.13 Stroke Limit and Emergency Stop

- **Relevant Signal**

Type	Sign	Meaning	Pin-out	PLC state	CNC diagnosis
Input signal	ESP	External Emergency Stop signal	CN61.6	X0.5	

- **Control parameter**

0	1	7
Corresponding F signal		

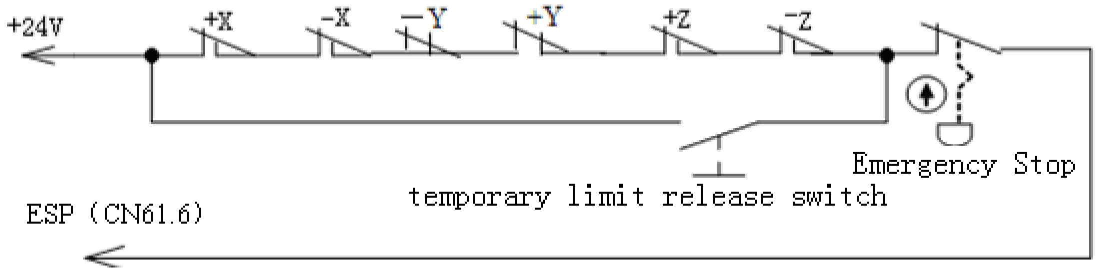
				MESP			
				F210.3			

MESP =0：The external emergency stop function active。

=1：The external emergency stop function is inactive。

- **External connection for machine**

The external connection method of emergency stop and stroke switch is as follows: (take 3 axes machine for example)



- **Logic control**

When the contact of emergency stop switch is closed, contact signal between ESP and +24V is cut off, and the CNC emergency stop alarm is generated. Meanwhile the CNC ENB signal is disabled and the pulse output is stopped. Other functions can be defined by PLC other than above functions by NC。

5.3.14 Tri-color Indicator

- **Relevant Signal**

Type	Sign	Meaning	Pin-out	PLC state	CNC diagnosis
Output signal	CLPY	Yellow light	CN62.31	Y2.2	
	CLPG	Green light	CN62.32	Y2.3	
	CLPR	Red light	CN62.33	Y2.4	

- Function description

Yellow light（Normal, no-running, no-alarm）, Green light（auto-running ）, Red light（system alarm）

5.3.15 Reset and Cursor Return

- **Relevant Signal**

Type	Sign	Meaning	Pin-out	PLC state	CNC diagnosis
Input signal		Reset key signal on MDI panel		X24.0	

- **Control parameter**

K0010								RESB

RESB =1: Reset and cursor return is active

=0: Reset and cursor return is inactive

APPENDIX

- Function description

When RESB of K10 is set to 1, by pressing the reset key in auto mode, the system reset and cursor returns to the beginning of a program.

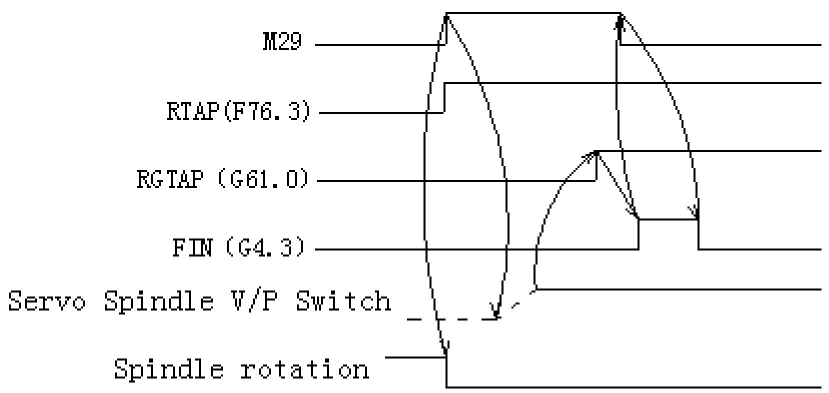
5.3.16 Rigid Tapping

- **Relevant Signal**

Type	Sign	Meaning	Pin-out	PLC state	CNC diagnosis
Input signal	VPO	spindle V/P output signal	CN15.6	X5.1	
Command input	M29	rigid tapping signal			
Output signal	VP	spindle V/P switch signal	CN15.20	Y5.0	
	TAP	TAP signal	CN15.21	Y5.1	

- Function description

When M29 is executed, VP signal is output and servo spindle switched form velocity to position, and then servo spindle VPO signal is output. PLC received the signal and set G61.0 to 1. The execution of G61.0 is finished and Y5.1 and G61.0 output simultaneously. The sequence is as follows:



5.3.17 Spindle Exact Stop

- **Relevant Signal**

Type	Sign	Meaning	Pin-out	PLC state	CNC diagnosis
Input signal		spindle exact stop signal		X25.7	
Output signal		spindle exact stop indicator		Y21.3	

- Function description

When 4th and 5th axis is valid and the corresponding state parameter RCS4/RCS5 is 1, by pressing spindle exact stop key in EDIT, REF, STEP/MPG, MANUAL modes, CS control mode can be switched.

5.3.18 External MPG control

- **Relevant Signal**

Type	Sign	Meaning	Pin-out	PLC state	CNC diagnosis
Input signal	EHDX	external MPG X choosed	CN31.5	X6.0	
	EHDY	external MPG Y choosed	CN31.6	X6.1	
	EHDZ	external MPG Z choosed	CN31.8	X6.2	
	EMP0	external X1 rate	CN31.9	X6.3	
	EMP1	external X10 rate	CN31.22	X6.4	
	EMP2	external X100 rate	CN31.23	X6.5	

- Function description

Standard ladder diagram supports external MPG with 3 axes. External MPG PSG-100-05E/L and ZSSY2080 are suitable. Refer to materials related to MPG for connection.

5.4 Standard Ladder Diagram

APPENDIX